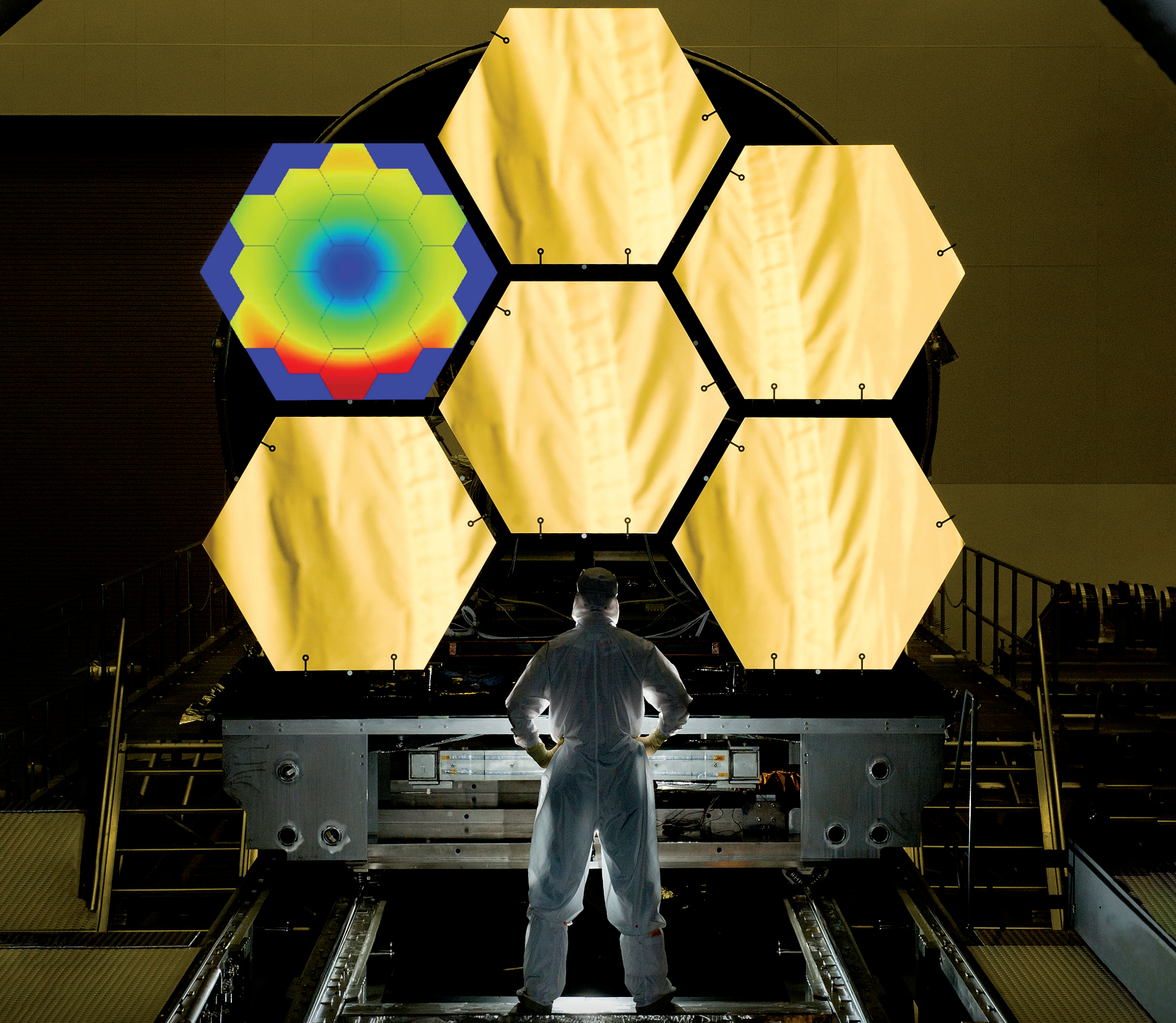
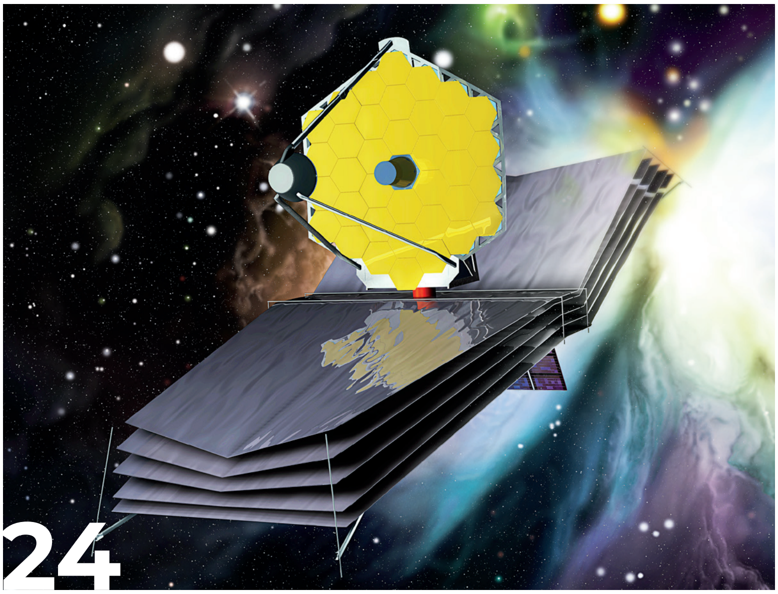
Ansys
ADVANTAGE
EXCELLENCE IN ENGINEERING SIMULATION
ISSUE 1 / 2022

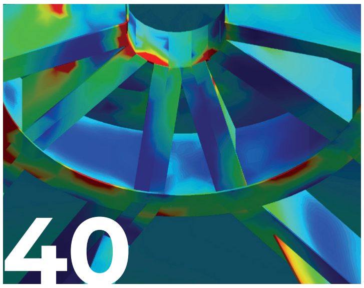


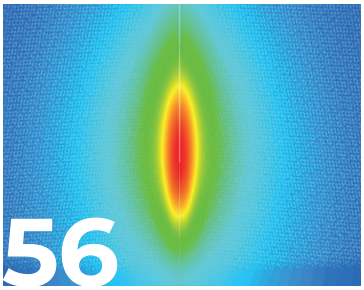
SIMULATION:
Solve the Mysteries of the Universe

Ansys

Table of Contents







Focus on Optimization

ADVANTAGE

Welcome to *Ansys Advantage*! We hope you enjoy this issue containing articles by Ansys customers, staff and partners.

The Editorial Staff, *Ansys Advantage*
ansys-advantage@ansys.com

Editorial Advisers
Lynn Ledwith, Mary Beth Clay

Executive Editor
Jamie J. Gooch

Managing Editor
Tim Palucka

Copy Editor
Abby Humphreys

Editors
Erik Ferguson

Editorial Contributor
Ansys Customer Excellence North America

Art Director
Ron Santillo

Designer
Dan Hart

ANSYS, Inc.,
Southpointe, 2600 Ansys Drive, Canonsburg, PA 15317

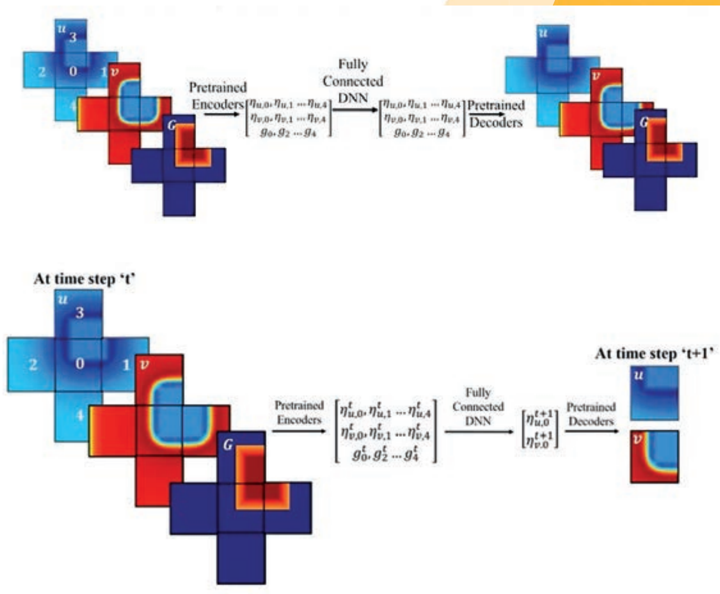
Subscribe at ansys.com/subscribe

AI Enhances Simulation Speed and Accuracy

By **Prith Banerjee**, Chief Technology Officer, Ansys

There has always been a trade-off between speed and accuracy, but when it comes to simulation, the use of artificial intelligence/machine learning (AI/ML) is tipping the balance to provide rapid innovation and higher confidence. By augmenting simulation methods with AI/ML, we have seen a 40x increase in speed on some applications, and that's just the beginning.

Simulation is used to analyze increasingly complex multiphysics and system-level phenomena across sectors, including challenges in next-generation products that are advancing electrification, autonomy, 5G, and personalized medicine. It enables our customers to solve problems that were once unsolvable and embrace virtual prototyping, which helps save time and money while increasing quality. However, while simulation is many times faster than physical testing and prototyping, the market demands even faster innovation, which Ansys has consistently delivered over the past 50+ years.



Steady-state (top) and transient (bottom) flow solvers for the Navier-Stokes equations focusing on small regions at a time.

COMBINING AI AND SIMULATION FOR FLUID FLOWS

When you are solving large flow problems with Navier-Stokes equations, it's complicated to solve for large regions via finite-element or finite-volume methods. You can increase the speed by focusing on small patches of the whole, then learning how those patches connect to understand the larger regions. That's where ML comes in. We can create an algorithm to take a single, small solution patch and move that across different regions to solve the larger flow model.

Using this steady-state flow solver as a base, we can transform it into a transient ML solver using the same method of focusing on small regions to speed up the solution and predict how the flow will vary over time. So, we have our methodology for ML-based steady-state and transient flow solutions. Now, we need to train it.

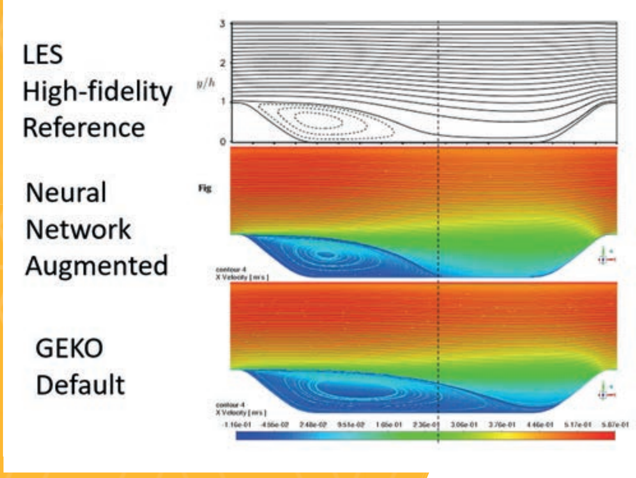
What we've done is taken the ML-based flow solver and integrated it with our Ansys Fluent solver to create a solver-in-the-loop. In essence, this enables massive training of data-driven neural networks for a variety of fluid variables.

Some initial results of this ML integration with Fluent on both fluid flow through a pipe and external aerodynamics for a car show a 30-40x faster time to solution using a single CPU.

TURBULENCE MODEL TUNING WITH AI/ML

Another example of combining ML and simulation is turbulence modeling. The most accurate solver for turbulence is the direct numerical solution (DNS), but that takes a tremendous amount of time, so some engineers use an approximation called large-eddy simulation (LES), which is faster than DNS and still quite accurate. An even faster approximation are Reynolds-average Navier-Stokes (RANS) equations, which are what Fluent uses. RANS equations are not as accurate as LES, so we use generalized k-omega (GEKO) models to get to the accuracy levels of LES while taking advantage of the speed benefits of RANS. However, identifying the GEKO parameters requires the specialized knowledge of a simulation expert. This is where we are applying ML.

Using ML-based methods, we tune the GEKO parameters automatically instead of manually to get to the accuracy of the LES model.

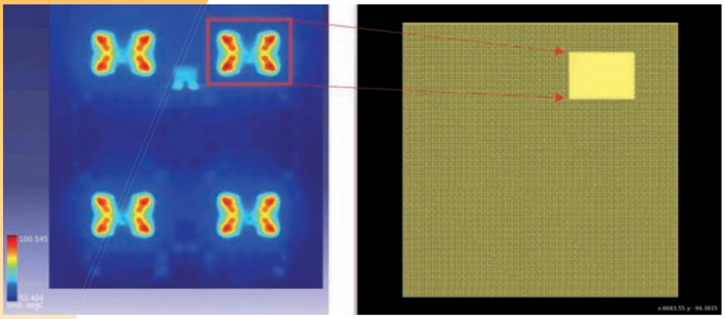


Turbulence modeling using large-eddy simulations and Reynolds-average Navier-Stokes (RANS) equations, with generalized k-omega (GEKO) models.

FIND THE BEST PLACE FOR FINE MESHES

Another example of using ML to identify critical locations to focus on to accelerate simulation can be found in the electronics field. When simulating electronic chip packages consisting of integrated circuits (ICs) on printed circuit boards (PCBs) or stacked as 3DICs, simulating their thermal properties at the most accurate level can take a long time. Typically, people speed this up by using adaptive meshing techniques. The time savings are significant. On a 10x10 chip-thermal method simulation, the run time can be reduced from 4.5 hours to 33 minutes using adaptive fine meshing.

You can think of a mesh as a grid of information used to simulate a particular phenomenon. A finer mesh contains more information and takes longer to solve but is more accurate. Ideally, you would only use finer meshes where they are needed most on chip "hot spots," with coarser meshes everywhere else, to obtain fast, accurate results. Using ML, we can automate the detection of chip hot spots to apply fine meshes where they are needed by predicting the temperature decay curves.

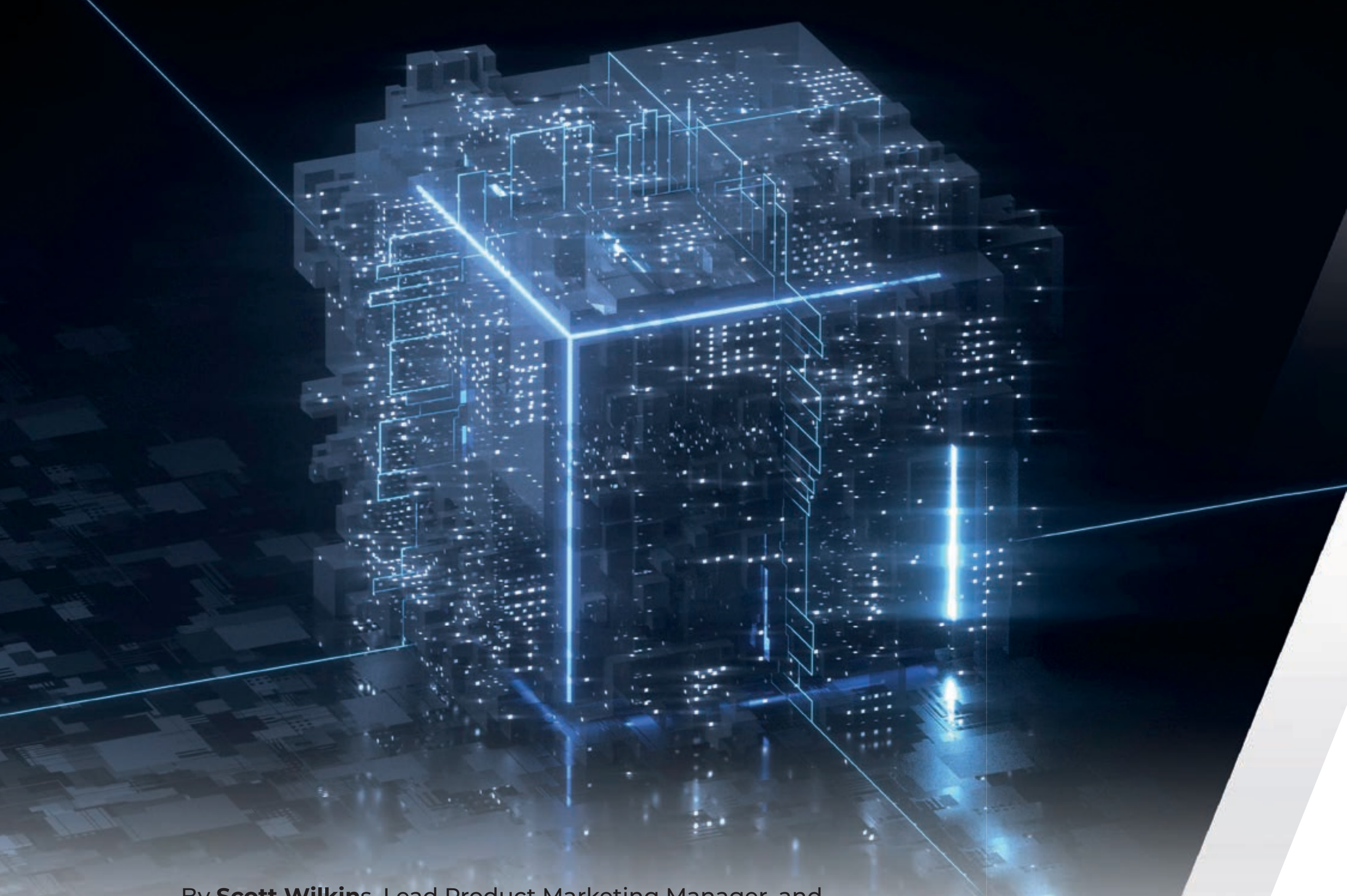


Using ML to automate the detection of chip hot spots by applying fine meshes where they are needed.

If you integrate these two concepts — adaptive meshing and automated hotspot detection — you can save significant time. For example, on a large 16mmx16mm chip, even a 200x200-micron coarse mesh takes 17 minutes to run. Using the ML-based adaptive meshing to place fine, 10x10-micron meshes where they are needed, it runs in 2m 40s. Both accuracy and speed are improved.

This is a just a sample of the ways our customers are using AI/ML to accelerate their simulations without sacrificing accuracy. Others are using ML to increase the accuracy of digital twins and speed complex, concurrent multiscale structural simulations of composite structures. Maybe your engineering team is working on a new application right now. We hope to read about your success in the near future.

Increasing Additive Manufacturing Build Success With MACHINE LEARNING



By **Scott Wilkins**, Lead Product Marketing Manager, and
Sak Arumugan, Lead Product Manager, Ansys

Additive manufacturing (AM) has made it possible to manufacture products with complex shapes that could not be manufactured using traditional methods, such as casting, molding, forging, or machining. By precisely depositing layers of powders (metals or polymers) and thermally fusing the powder particles of each layer to themselves and to subsequent layers, there is almost no limit to the geometries of real-world working parts that can be produced. From small heat sinks that are optimized for efficiency to enormous rocket engines that will launch future astronauts into space, AM can do it all.

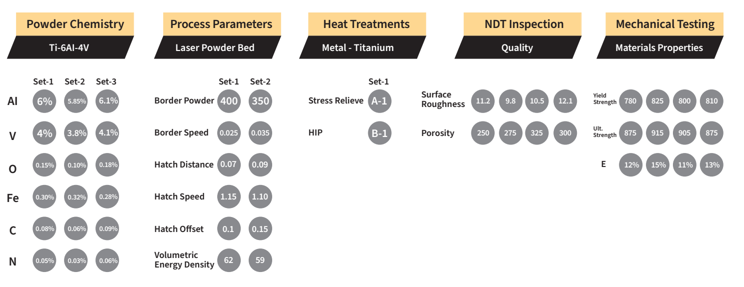
What AM couldn't do until now is produce these parts as quickly and consistently as manufacturers would like. Additive manufacturing typically requires numerous rounds of trial and error to optimize a new additively manufactured part. Even when the design and AM process have been finalized, other variables can be introduced into the process — such as a new batch of material or different environmental conditions — that can affect repeatability. Considering the high cost of metal powders that are wasted in a failed build, additive manufacturers cannot afford failures.

THE CHALLENGES OF ADDITIVE MANUFACTURING

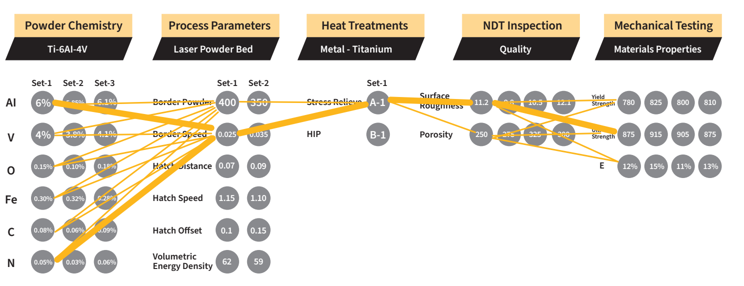
The key challenges for large-scale adoption of AM in industry are speed and reliability. How can you rapidly optimize process parameters for additively manufactured parts, thus reducing time to market? Overcoming these challenges is made even more difficult because every AM process has its own set of process parameters, including material properties, manufacturing settings, and changing environmental conditions.

Even just focusing on material properties can result in a large number of variables. The powder chemistry of an alloy can vary widely because of small levels of impurities that can change the composition of every batch. What percentage of variance in the concentration of titanium in a Ti-6Al-4V alloy powder can be tolerated before an AM build fails? How do levels of impurities such as oxygen, iron, carbon, and nitrogen affect the tensile strength of the resulting AM part? Possible variations in processing parameters in a laser-powered AM machine (border power, border speed, hatch distance, hatch speed, hatch offset, volumetric energy density, etc.) add to the complexity of the AM challenge. Add in heat treatments, non-destructive testing (NDT) inspection, and mechanical testing variables, and soon you have a major data management challenge on your hands.

You may also encounter challenges in predicting cause-effect results when one or more variables change during the build process. Being able to predict what will happen plays a large role in ensuring a high rate of build success when so many process-property combinations exist.



A representation of the sheer number of AM variables

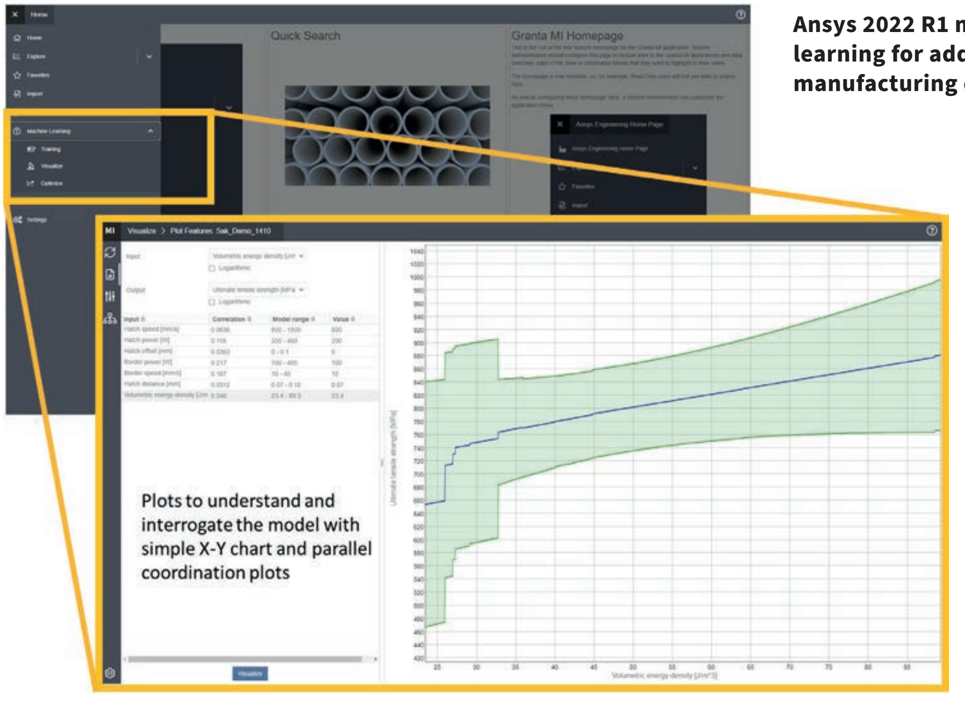


Using machine learning to identify the causal links between AM data

A DATA-DRIVEN APPROACH TO HANDLING AM DATA

The first step to confidence in the AM process is helping engineers capture and manage their materials information. That's the primary focus of Ansys Granta MI. It enables companies to collect, organize, and store their own materials data gathered from testing of incoming raw materials and outgoing end products. This customized data is invaluable in fine-tuning the AM process for a company's proprietary materials intellectual property. Granta MI also gives companies a single repository for materials information — a "single source of truth"— which avoids the problem of different engineers working with different data.

Even with the data management capabilities of Granta MI, real-world AM datasets are typically sparse and noisy — "sparse" meaning that there are gaps in certain lines or columns of a dataset, and "noisy" meaning that there can be a large spread between different measurements of the same property. Another problem is that even though there may be gaps in the data, these material datasets can become too large to efficiently extract useful data.



Ansys 2022 R1 machine learning for additive manufacturing data

So, Ansys partnered with Intellegens to incorporate its Alchemite™ algorithm to statistically fill in the gaps and reduce the noise in the data. Machine learning (ML) algorithms, often based on neural network approaches, build models from sample data, known as the training data. Based only on what they have "learned" from this data without further explicit programming, these models support tasks including:

- Predicting missing values in the rest of the dataset
- Predicting outputs for a new set of input variables
- Optimizing performance by predicting which inputs will best achieve desired outputs
- Identifying outliers, anomalies, or clusters in the data
- Identifying which inputs are strong drivers of which outputs

Machine learning uses existing AM project data to build predictive models that identify key process-property relationships, guide your testing program, and propose optimal processing parameters. In short, Alchemite can extract value from your data even when it is sparse, noisy, and in high volumes. When you have too little data, ML helps you focus your data acquisition efforts efficiently by identifying key parameters that are missing. When you have too much data, Alchemite can extract the key parameters that are critical to predicting AM build success. Learn more about how the Ansys solution works for AM data with our on-demand webinar (ansys.com/events/ansys-2022-r1-additive-manufacturing-data-update).

USING MACHINE LEARNING TO IDENTIFY THE CAUSAL LINKS BETWEEN AM DATA

This new level of insight into AM data enables engineers to control the AM process and optimize material and part performance. You can achieve results with a greatly reduced number of experimental test cycles, which helps accelerate time to market and leads to substantial end-to-end total cost savings for AM programs.

A Solution in Three Apps

To make the software easy to learn and use, Granta MI for AM solutions has been distilled into three apps: a training dataset app, an optimization app, and a visualization app.

1. The **training dataset app** helps engineers create the training dataset for the neural network. With just a few clicks, you can extract key data from the Granta MI database to form a complex feature matrix that can be used to train the neural network.

2. Using the **optimization app**, you can take the model created by the neural network and interrogate it to get answers to your specific questions regarding AM processes. Questions such as "How can I maximize a certain property?" or "How can I reduce a specific failure mode in my build?" can be answered by this app.
3. Finally, the **visualization app** lets you quickly understand your data. A graph can be much easier to understand than a complex table of data points. In the visualization app, engineers can take a slice of a complex dataset and have it displayed graphically to generate insights from the data.

BRIGHT OUTLOOK FOR MACHINE LEARNING IN ADDITIVE MANUFACTURING

Almost 60% of 375 businesses surveyed by MIT Technology Review in 2016 were incorporating ML into their operations. A report by the leading marketing research firm PwC notes that 100% of companies identified as "digital champions" (as opposed to "digital novices") are planning to use ML by 2025.

They have good reasons for doing so. According to Ansys' calculations, AM processes that combine a data management system with ML are predicted to result in a 50-90% reduction in the number of experiments needed to establish the correct AM processing parameters. This and other benefits will lead to a 10% cost reduction in the global AM market. Learn more from the Ansys white paper "How Machine Learning Helps Getting Additive Manufactured Parts to Market Faster" (ansys.com/resource-center/white-paper/granta-machine-learning).

For AM to continue growing as an industrial manufacturing process, manufacturers will have to rely on improvements that reduce the number of build failures and give them confidence in their workflows. They can't afford to keep wasting money on expensive metal powders that only end up on the scrap pile. Ansys will continue to add features to Granta MI and work with Intellegens to make ML an even more important part of the AM build process, ensuring that the benefits of additive manufacturing will be available to future generations of engineers.

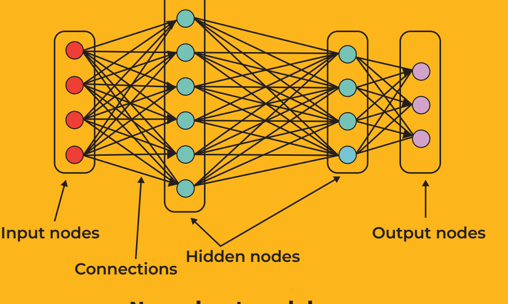
If you would like to see how this can work for your company, book a demonstration with our team: ansys.com/products/materials/arrange-demo.

HOW A NEURAL NETWORK WORKS

Neural networks are extremely valuable in additive manufacturing (AM) because they enable engineers to make sense of complex, multidimensional data, finding relationships that cannot be easily understood by human observation or through conventional statistical or analytical techniques.

In a neural network, system inputs are captured by input nodes and used to compute the values of hidden nodes in the next layer using nonlinear equations in which weightings control the contribution of each input. This process repeats for each layer of the network until outputs are predicted. The machine learning (ML) algorithm takes the inputs from a training dataset and adapts the weightings throughout the network to minimize the difference between predicted and actual outputs for the training data. The resulting model can then be used to predict outputs for new inputs.

Alchemite also includes an advanced method called "non-parametric probability distributions" to quantify uncertainty when predicting data. This provides valuable guidance on data quality and on which predictions will have the highest probability of success.



Neural network layers

Make Better Contact

Bu **Jay Pathak, Director**, Research and Development, Ansys

While Ansys has already integrated artificial intelligence (AI) and machine learning (ML) into many of its software solutions, we continue to invest in research to move these technologies forward in the field of simulation. As Ansys CTO Prith Banerjee discussed in his editorial (see page 4), we are applying AI/ML on many fronts to speed up Ansys solvers while maintaining accuracy. I'd like to share some behind-the-scenes details of our R&D efforts around deep learning and how they relate to increased solver speeds.

Recently, I co-authored a paper with Ansys Senior R&D Engineer Rishikesh Ranade that's been accepted for publication by ASME's *Journal of Mechanical Design*. The paper describes the capabilities of a new 3D architecture based on convolutional neural networks, or CNNs, a type of deep learning network typically applied to analyze visual imagery. Called ActivationNet, this architecture may soon be incorporated into Ansys structural and fluid solvers to improve the quality of detecting 3D surfaces in contact with one another.

Many structural problems involve accurately replicating how 3D surfaces interact, so physical phenomena resulting from that contact — such as stresses, fatigue, and wear — can be confidently predicted. The challenge? Setting up high-quality contacts for a large assembly of bodies is very manual. While algorithms exist for automatic contact detection, they are not foolproof, and they often end up over-detecting or under-detecting contacts when dealing with assemblies that combine both large and small parts. In addition, contact detection during the non-linear solution of large assemblies is time-consuming. At Ansys, we are always looking for ways to improve speed within our solvers. Deep learning, and 3D CNNs in particular, are great at detecting local features in images — which inspired us to investigate their use for surface-to-surface contact detection.

ENSURING HIGH-QUALITY AUTO CONTACT DETECTION BEFORE SIMULATION BEGINS

Because it's impossible to fully replicate every physical detail of today's intricate product designs in a time- and cost-effective manner, 3D surfaces are currently modeled using point clouds — a set of data points in space — or surface meshes that approximate the product's geometry in a manner that's numerically manageable. Auto contact detection algorithms work directly using surface mesh pairs, using features such as proximity/gap between surfaces, the angles between surfaces, and the overlap or intersection of surfaces.

However, because contact pairs are detected based on deterministic algorithms — with a single gap tolerance representing contacts for all interacting bodies — the result can be "bad contacts." For example, surface pairs can have a large gap, some surface pairs might be perpendicular to others, or there may be a large angle between different surfaces. And, unfortunately, engineers have no easy, automated way to distinguish "good" contact models from lower-quality ones that should be improved before the simulation run begins. This process is very manual and time-consuming because engineers need to look at each pair, then accept or delete them.

To address this challenge, Ansys has developed a proprietary algorithm, ActivationNet, that assigns a quality score to each surface-contact model in the product design before any simulation is conducted. Engineers can flag these lower-quality surface-contacts and delete them before simulation to deliver accurate results.

ActivationNet uses coarse point-based or mesh representations of contacting surfaces to generate a series of activation states on computational grids of different resolutions. The activation states are based on the spatial location and density of points in a normalized 3D space. These states are simply markers assigned to each pixel of the computational grid to represent the individual contacting surfaces, as well as the overlapping regions between them. These activation states are fed into deep neural networks that extract features equivalent to the "handcrafted" inputs — for example, surface proximity and angle — that were initially defined by human engineers.

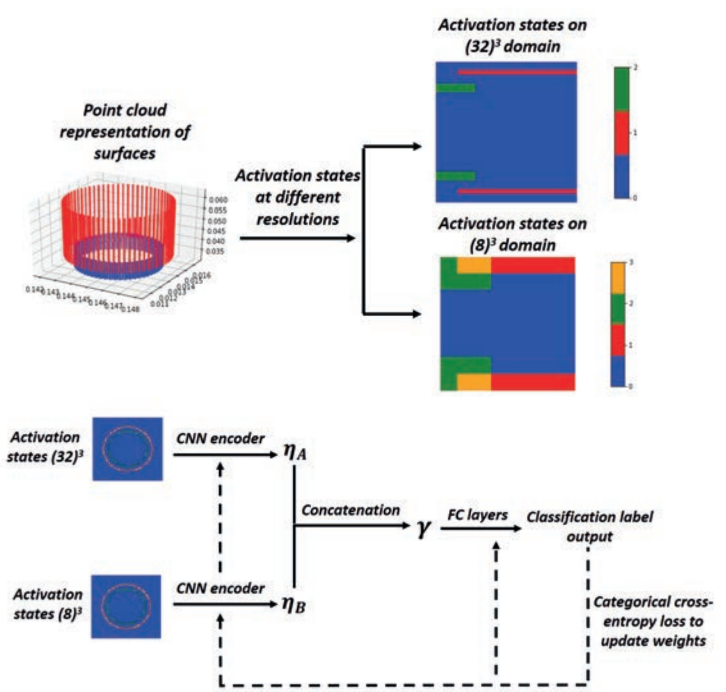
Because these neural networks work in probabilistic space, they exceed the capabilities of deterministic algorithms and rough meshing techniques to represent important and intricate characteristics of 3D surfaces — and they produce more accurate assessments of how well surface interactions are modeled. These networks "learn" the characteristics of good surface-to-surface contact interaction and can accurately rate the quality of each interaction in the product model.

ActivationNet computes a score ranging between 1 and 100 for every pair of 3D surface contacts in the product model. Surface-interaction models with a score of 90 or above can be trusted to deliver accurate simulation results, while interactions with scores below 20 can be safely deleted. Others that score between 20 and 90 should be reviewed or remodeled for complete confidence.

ActivationNet will help engineers improve productivity when setting up contacts before running simulations, as well as increase the accuracy of their ultimate results.

The Ansys software development team is running exhaustive tests to prove ActivationNet's ability to assess and rate the quality of 3D surface-contact modeling for Ansys customers. The team's research has confirmed that ActivationNet is learning true representations of interacting surfaces and performing better than existing deterministic algorithms at this sophisticated task. The results from ActivationNet have been shown to agree well with the expected results for various interacting surfaces with arbitrary shapes and sizes.

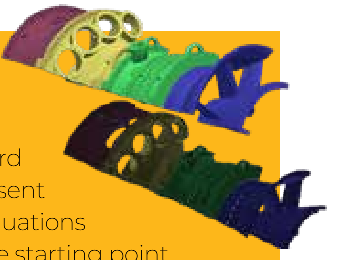
The goal is to help Ansys customers in every industry master the art and science of modeling 3D surface interactions — a fundamental application that will help improve simulation results and product confidence.



The ActivationNet algorithm uses the spatial location and density of points in a cloud or mesh to generate activation states (top). Depending on the problem at hand, these activation states reflect the geometric model at a range of resolutions. The collected activation states, which represent contact-surface interactions, are handed off to neural networks for assessment (bottom).

RESOURCE

"ActivationNet: Representation Learning to Predict Contact Quality of Interacting 3D Surfaces in Engineering Designs," *Journal of Mechanical Design*, Rishikesh Ranade and Jay Pathak, March 18, 2022. tinyurl.com/2m7c7tyt



THE SHAPE OF THINGS TO COME

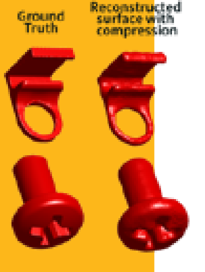
A team of Ansys researchers recently co-authored a paper with Stanford University on how machine learning (ML) might be applied to represent geometries in a manner that is suitable for learning partial differential equations used in various Ansys solvers. Lightweight geometry representation is the starting point for any machine learning closer to the solver's realm.

Capitalizing on convolution neural networks, the Ansys-Stanford team is leveraging new, data-driven, and physics-informed machine-learning models that could allow computer-aided design (CAD) engines to quickly represent simple shapes via a new geometry encoding process. The new, lighter-weight encoding process results in a compressed spatial representation that still produces numerically accurate results.

While progress is being made, we continue to invest here to make it more scalable, so it can quickly represent any level of complexity of shapes required by Ansys customers. Eventually thousands of modeling applications, across many complex shapes and very intricate geometries, could potentially be simplified and accelerated with these new geometry encoding techniques.

Ultimately, the Ansys R&D team is working to make the geometric understanding of Ansys solutions smarter and more autonomous, able to intuitively mix lightweight and heavier-weight representations of 2D and 3D objects, as needed, to solve every customer problem.

LEVERAGING AI AND ML TO CREATE LIGHTER-WEIGHT GEOMETRIC REPRESENTATIONS



We know artificial Intelligence (AI) and ML can solve extremely large, numerically intensive, complex problems in computer vision and natural language processing, and we are excited about applying it for learning physics, which in many ways is similar. The recent paper co-authored with Stanford focuses on improving the state of the art in representing geometries in a manner that will be suitable for learning physics down the line. We have shown strong generalization of our model including verification from other public datasets.

The model, although trained with simple parts (such as sphere, cylinder, or prisms) and unions or intersections with other similar parts, generalizes to complex parts used by Ansys customers such as engines, turbines, etc. Instead of relying solely on human software developers to advance the state-of-the-art of engineering simulation, we're exploring the potential of machine learning to complement our traditional software development efforts.

The performance of the processor on more complex 3D geometries shows that the compressor produces almost the exact same results.

By identifying repeating patterns in geometries, machine learning can encode only relevant information and hence allow a decent level of compression to represent geometries. A trained model can be used to decode back this representation when needed into full-fledged 3D or 2D geometries.

The broad adoption of AI and ML is having enormous implications for the performance of many products — and incorporating this advanced technology is making the job of product development teams more complex and challenging than ever. At the same time, Ansys is capitalizing on those same AI and ML capabilities to help customers solve their most complex problems. We're starting with the task of geometric representation, because that's a foundational problem that affects virtually every engineering team. But there's no limit to other applications of ML, such as in solving physics, meshing, identifying different domains, etc.

For everyone in our industry, it's an exciting time, and I think we're only at the beginning of realizing the potential of artificial intelligence and machine learning to change the world as we know it — including the capabilities of simulation software.

RESOURCE:
"Geometry encoding for numerical simulations," Amir Maleki, Jan Heyse, Rishikesh Ranade, Haiyang He, Priya Kasimbeg, Jay Pathak; April 2021. researchgate.net/publication/350964239_Geometry_encoding_for_numerical_simulations

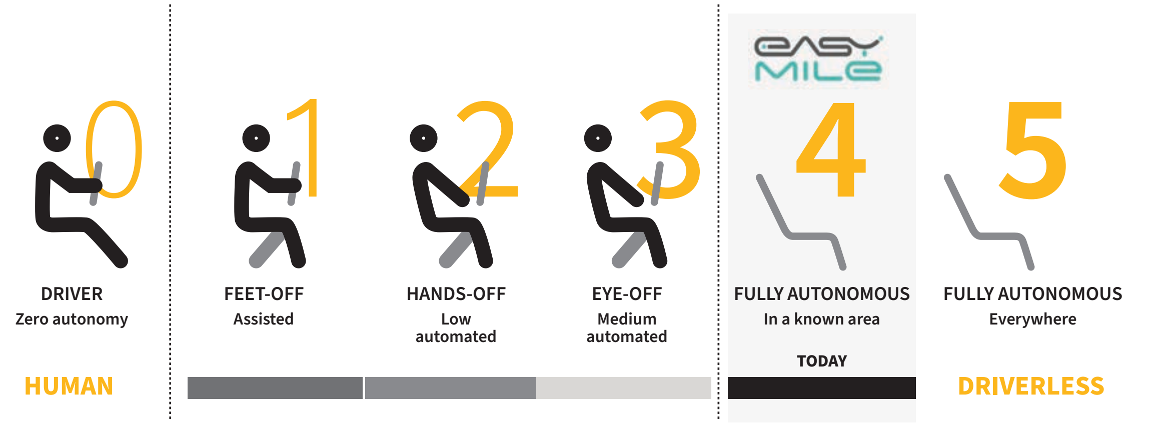
Paving the Way for Safety in Electric Autonomous Vehicle Technology

By **Laura Carter**, Staff Writer, Ansys Advantage

Today, fully electric driverless shuttles are rolling through college and business campuses and winding through city streets to shape the future of public and private transportation. All-electric driverless tractors are also enabling autonomous material handling in manufacturing environments to transport materials seamlessly, without interruption, speeding time to market. With the advent of these artificial intelligence (AI)-powered vehicle systems comes a host of safety concerns, which is why Ansys medini analyze was chosen by all-electric autonomous technology supplier EasyMile to define and demonstrate the safety of their electric driverless solutions before they ever see pavement. Ansys and CADFEM France worked together to provide the software and support EasyMile needed to achieve their goals.

EasyMile delivers complete solutions for driverless mobility and goods transportation, partnering with blue chip original equipment manufacturers (OEMs) to autonomize their electric vehicles. The company started out in passenger transport with the EZ10 shuttle, which remains a key solution for them, and they've applied this technology to an autonomous TractEasy tractor. The tractor is designed for efficient material handling in manufacturing environments, optimizing logistics operation through cost reduction and increased productivity. Operating at Level 4 (L4), EasyMile has completely removed any human supervisor on board both the EZ10 and the TractEasy during vehicle operation.

The company's early success is evident in the number of recent EasyMile vehicle



EasyMile has 10 Level 4 autonomous vehicles deployed around the world.

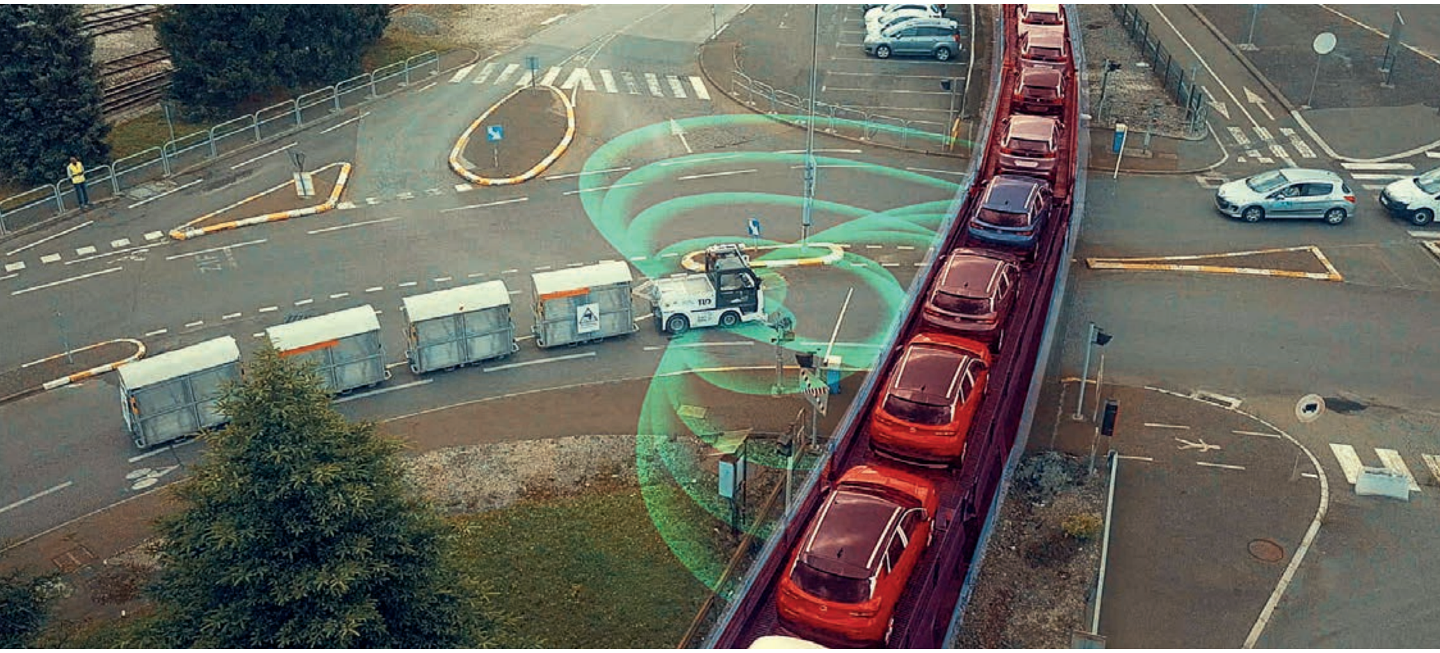
deployments at Level 4 around the world. They're at 10 deployments and counting, which EasyMile says is the largest L4 vehicle deployment on the market to date. A unique in-house software package with an embedded system helps automate both the EZ10 and the TractEasy transportation platforms. The software coordinates with numerous vehicle sensors to collect data on the surroundings, which is shared with EasyMile to provide localization, perception, and path planning, resulting in more automated, less infrastructure-centric environments.

MEDINI ANALYZE DRIVES COMPREHENSIVE VEHICLE SAFETY ANALYSIS

Autonomous vehicle operation depends on a high level of information to operate safely. Both the EZ10 and the TractEasy are equipped with a full range of lidars, radars, sensors, cameras, and GPS hardware, along with navigational software that collects and processes data to get 360-degree views of the environment. All use real-time vehicle data processing that will inform autonomous function and intelligence via a software-embedded system. The system is designed to automate transportation platforms. For people transport, the goal is passenger safety, as well as the safety of drivers and pedestrians in the surrounding environment. For goods and material transport, there are predetermined standards for safety already in place that must be tweaked for the safe operation and control of an unmanned vehicle. For both applications, the challenge for EasyMile is working with new systems involving a high level of complexity, without clearly defined

CO-CREATING A GREENER FUTURE

In the race to net-zero emissions, electric driverless technology presents many exciting possibilities for people, passengers, and the environment. Electric autonomous vehicles have the potential to democratize transportation, anticipate and coordinate traffic patterns, reduce CO_2 emissions, and free up green space. Looking toward the future, EasyMile is working behind the scenes with industry nonprofits and academics to launch a European chapter of Partners for Automated Vehicle Education (PAVE). The group is a diverse coalition that unites industry partners and nonprofit members, including sustainability advocates, who believe in the potential of autonomous vehicles to make roads safer, more accessible, and more environmentally friendly than they are today. EasyMile is also heavily engaged with the French government from their base in Toulouse and with government officials from other countries, including the U.S. and Germany, to help meet these objectives. The technology provider is very active in the electric autonomous technological space and is increasingly involved in global conversations that will drive sustainability in the future. Of course, EasyMile wants to commercialize their solutions, but they also see themselves as playing an important role in developing the regulatory framework and government initiatives around their electric autonomous vehicles. With an assist from Ansys, EasyMile is helping to facilitate an understanding of electric autonomous vehicle technology among officials that will encourage more deployments in the future.



The EasyMile TractEasy is an electric autonomous tow tractor solution for material handling in both indoor and outdoor environments.

methods for how to demonstrate the safety of their autonomous vehicle applications. This is where medini analyze came into the picture.

"We were developing a very complex autonomous system and needed to demonstrate its safety," says Romain Dupont, Safety Manager at EasyMile. "This required our team to take a different approach using new solutions to manage requirements, to manage simulation, and to manage a lot of activities that compose the development life cycle. Among these, medini was key to helping us reach our goals."

Autonomous vehicle deployments involve education sessions with emergency responders. They also require EasyMile to work with clients globally to help educate government officials at the federal, state, and local levels so vehicles are deployed with all regulatory approvals. Ansys medini analyze provided EasyMile with a turnkey system with all the necessary tools at their disposal in one solution. The software enabled EasyMile to manage and define safety analysis of their autonomous products using their own process within medini analyze, thanks to the software's flexibility to adapt to this process. At the vehicle level, EasyMile used the software to conduct all the safety analyses required from their customers and provide the documentation needed for European safety regulators, such as the Safety Services Organization (SSO). It was also used at the subsystem level to analyze connected infrastructure, including interactions among the operator, the deployment team, and all other stakeholders in the transportation system. Recently, with the help of the Ansys team, EasyMile successfully added another tool to medini analyze (a custom extension of the software) to help them drive all safety concepts and requirements into one model for their passenger and material transport applications. The tool supported their product line engineering approach and can generate the documentation of all their products for every platform.

"I would say it's almost like we're co-constructing two things here," says Benieke Terverton, Head of Communications at EasyMile. "We've got our internal approach, as well as what we offer to our clients and our partners about the safety of our products. Ansys and medini analyze have allowed us to co-construct those two approaches to offer something new to the world of autonomous vehicles. We deal with a lot of different assessors and regulators around the world. With Ansys, we can manage these interactions successfully."

ADDITIONAL RESOURCES:

ansys.com/products/safety-analysis/ansys-medini-analyze

ansys.com/technology-trends/autonomous-engineering

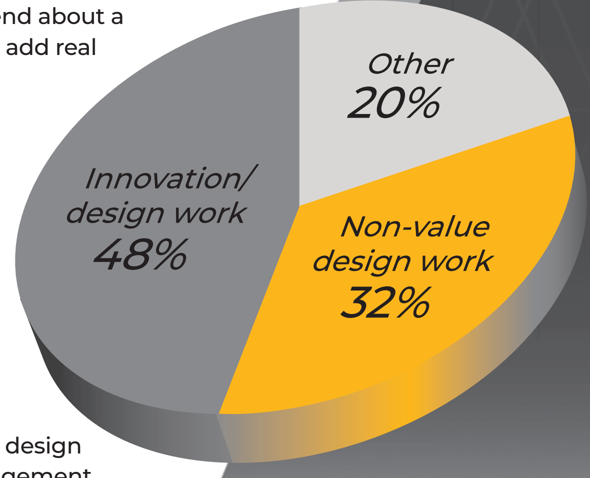
ansys.com/technology-trends/electrification

Optimized Processes Drive an OPTIMAL RETURN

by **Priyanka Gotika**, Senior Product Marketing Manager, Ansys

Product development has never been more challenging. As product designs have become more complex, so has the work associated with designing them and verifying their performance under a wide range of operating conditions. Engineers are faced with applying multiple physics at both the component and system levels, working against tight budgets and aggressive launch schedules, and collaborating across disciplines and functional boundaries to drive innovation.

With this level of complexity and pressure to deliver innovative products quickly, it might surprise you that engineers still spend about a third of their time on administrative tasks that don't add real strategic value.



Engineers spend a third of their time on non-value-added work.
Source: Tech-Clarity

The three major inefficiencies are:

1. Failing to address the hidden delays of the job submission queue
2. Ineffectively managing the steps in between simulations
3. Coordinating huge amounts of decentralized simulation data

A PURPOSE-BUILT SIMULATION WORKFLOW SOLUTION

To meet the challenges involved in a modern digital design process, a robust simulation and process data management (SPDM) strategy is a must. SPDM helps organizations build and maintain the digital thread of data connecting engineering to the wider enterprise. Ansys Minerva is an enterprise-level SPDM solution that is purpose-built to provide decision support and drive collaboration among your interdisciplinary team.

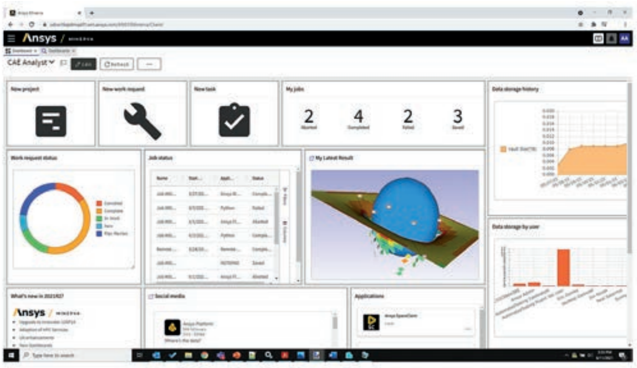
Minerva provides a central repository for your organization's simulation and process data, introducing cross-functional transparency and real-time visibility into the product development process. By automating process handoffs, optimizing workflows, and supporting proven best practices, Minerva significantly accelerates the development process without sacrificing accuracy or analytic rigor. Because the entire engineering team has access to the same data in real time, it becomes exponentially easier to weigh trade-offs, make fact-based decisions, and move one step closer to market launch every day.

Let's take a look at how Minerva solves the three challenges to simulation efficiency.

1. Addressing the Hidden Delays of the Job Submission Queue

Once your engineers have submitted their simulation jobs to the processing queue, when will the simulations start? When will processing be finished? And will you make your product launch deadline?

In a recent study conducted by Ansys and Digital Engineering, 21% of respondents said their most frequent simulations take at least nine hours to complete. Another 45% reported that their typical simulation runtime ranges from three to eight hours.



With Minerva, the entire team has access to the same data in real time, making it exponentially easier to weigh trade-offs, make fact-based decisions, and move closer to market launch every day.

Even when using some form of high-performance computing (HPC) cluster, shared resources are often overwhelmed. With multiple three- and nine-hour jobs lined up to use the same processing cores, average simulation times can easily double or triple once wait times are factored in.

Why not simply add more processing cores? Because IT wants to see a high rate of utilization, which translates to a high return on investment. Underutilizing a cluster of expensive computing resources is every executive's nightmare. So, virtually every engineering team — even at the largest companies — has to contend with a long job submission queue.

Minerva simplifies the HPC job submission process. It features a single, dedicated user interface that makes it fast and seamless for simulation users to create and submit jobs, monitor their status in real time, and understand the estimated end time of their results.

With Minerva, engineers can see up front how loaded the HPC cluster is, estimate their total wait/run times, and make informed decisions. Working collaboratively, the product development team might visualize the queue and make strategic choices about when, and in what order, to submit jobs to keep the shared launch schedule on track.

2. Managing the Steps In Between Simulations

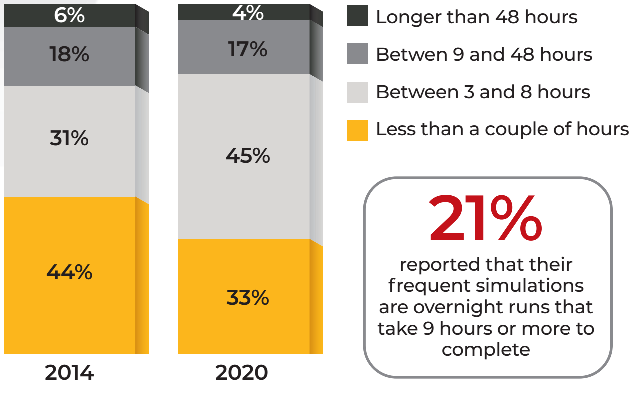
Even when the simulations run at the fastest possible speed with no glitches, the entire process can slow down if engineers have to manually interact with the process in between multiple simulation runs.

Many product development teams still rely heavily on manual processes to manage the broader workflows involved in simulation-driven product development. They attempt to manage the incredibly complex, enormously sophisticated, and strategically important process of product development via emails and spreadsheets. They waste time every day because of blind spots, bad handoffs, missed communications, lost data, lack of version control, unnecessary rework, incorrect assumptions, and other sources of inefficiency. Projects lag because no one knows what to do next or who is responsible.

As a result, engineers risk making a range of mistakes, like importing the wrong data, using an outdated file version, or re-running a simulation. There's no real accountability, so it's hard to keep track of where projects are in the pipeline. No one is sure what the next step is or who owns it. No one is even sure if the last step was completed successfully. Older projects lag, and new projects proceed slowly because past simulation results are pushed aside and forgotten instead of being re-used when a similar project is kicked off — resulting in redundant effort and lost time.

Task trees, role-specific views, and automatic notifications of project

How long do your most frequent engineering simulations take to execute?



Time simulations take to run, according to a study by Ansys and *Digital Engineering*

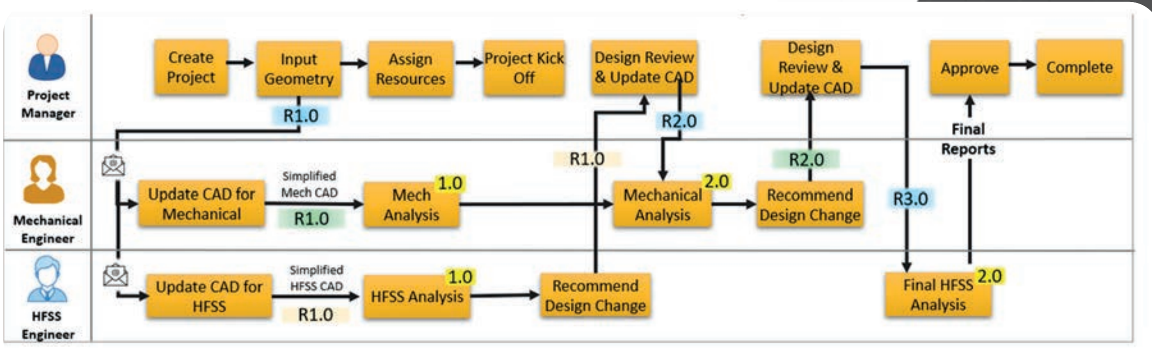
updates available in Minerva provide all users with real-time visibility, traceability, and accountability. Guesswork, errors, and tedious manual handoffs are replaced with speed and automation to support the fast, optimal execution of your simulation-driven product development processes. Minerva delivers all this functionality in an intuitive, easy-to-use package that resembles consumer software.

3. Coordinating Huge Amounts of Decentralized Simulation Data
Simulation-driven product development generates huge volumes of critical data. This data feeds all your most critical decisions, enabling you to make fact-based choices and trade-offs as you launch products into the market. But where exactly is it?

In most organizations, simulation data is hard to find. Some of it might be on a jump drive or in a desk drawer. Other data might be stored on the laptop of an employee who's out on vacation. Maybe it's stuck in an email inbox or on a public file-sharing website, lost in that gray space between engineering functions. The inability to centralize, share, search, and re-use this data is negatively impacting your product innovation, development costs, and time to market.

Your employees may not tell you this, but they could be spending

Bottom-up Customer View — a Day in the Life of a Simulation Engineer



Minerva provides a high-performance computing queue dashboard, proactive alerts, and traceability.

hours of their time every day trying to locate data. And, even when they find the simulation data files they're looking for, the confusion continues. There may be multiple versions with different owners and varying file names. No one is sure which file is the right one.

By creating a single, searchable, centralized repository for all your simulation data, Minerva maximizes speed, efficiency, cost effectiveness, and engineering productivity throughout the product development cycle. It promotes cross-functional, real-time transparency, traceability, and accountability for all simulation data. Minerva supports keyword searches at the metadata level, automatic version control, and user-specific dashboards that enable easy file access.

In addition, Minerva makes it easy and seamless to use your existing simulation data as an input for new product studies. Instead of having your employees "re-invent the wheel" with each product design iteration, you can begin with a foundation of existing engineering knowledge and build from there.

A TRACEABLE, UNDERSTANDABLE RESULT
After a simulation, users benefit from a visualization of the simulation workflows, traceability for all data, and an automatically generated report that has all the results files, including 3D images. With Minerva, everything related to the simulation is captured and shared in a centralized location instead of being saved locally, eliminating time-wasting searches for data. Any team member around the world can call up the results and be sure they are accessing the correct version.

From planning simulation job submissions to reviewing the results, having a purpose-built simulation process and data management solution like Ansys Minerva can save time, computer resources, and your engineers' peace of mind. Let Minerva take control of your complex simulations from beginning to end and reap the benefits of being first to market.

SIX PRACTICAL BENEFITS FUEL A FAST, SUCCESSFUL LAUNCH

While Ansys Minerva is powerful enough to manage the complexities of multiphysics simulations and interdisciplinary teams, it's also extremely practical — designed to address real-world problems and streamline everyday workflows. Minerva helps your engineering team by:

- **Establishing a searchable, centralized repository for all data.** By collecting real-time simulation data in one location for a single version of the truth, Minerva increases cross-functional transparency, accountability, and traceability. The entire development team can quickly assess simulation results, weigh trade-offs, and make fact-based decisions in a collaborative environment.
- **Enabling visibility and control of all simulation processes and workflows.** With Minerva, no one needs to wonder about current project status, next steps or who's accountable. Task trees, role-specific views, and automatic notifications keep the entire team informed, eliminating process lags and blind spots. Even the most complicated workflows and engineering processes can be automated and streamlined via Minerva.
- **Maximizing usage of shared computing assets.** Most product development teams share a high-performance computing (HPC) cluster, which can easily be overwhelmed by multiple large simulation jobs. Via Minerva's dedicated user interface, engineers can see cluster capacity and wait times up front, prioritize jobs and make informed decisions, and monitor projects in real time.
- **Providing real-time reports and dashboards.** Minerva's highly visual dashboards and easy-to-understand reports enable the strategic, top-level management of all simulation projects. Diverse stakeholders can see the project status in real time, so they can flag roadblocks, intervene, or reprioritize as needed, ensuring that all product launches are on track in terms of time and costs.
- **Creating an integrated ecosystem of product development technology.** Not only is Minerva designed for interoperability with Ansys simulation software, but it also integrates seamlessly with computer-aided design (CAD), computer-aided engineering (CAE), product lifecycle management (PLM), and other engineering tools that drive product development. Minerva's open, vendor-neutral architecture provides a common, best-in-class platform that connects and maximizes all technology investments.
- **Amplifying human resources by increasing engagement with simulation.** Minerva delivers powerful features and functionality via an intuitive, graphical interface that resembles consumer software. This ease of use makes Minerva accessible to nonexperts in engineering simulation, allowing more team members to initiate and manage lightweight, early-stage simulation runs. Important issues can be flagged before the team invests in larger, more comprehensive simulations managed by engineers.

Hitching a Balloon Ride to Space

By **Tim Palucka**, Managing Editor, Ansys Advantage

When Austrian skydiver Felix Baumgartner parachuted from a pressurized capsule attached to a helium balloon from about 24 miles in the stratosphere above New Mexico on October 14, 2012, most people just saw an incredibly dangerous stunt resulting in a big success for Baumgartner and his corporate partners. But the husband-and-wife team of Sohrab Haghighat and Saharnaz Safari saw an opportunity to start a business using balloons to lift small rockets into the air, where the rockets would be launched above the atmosphere and carry their cargo to a predetermined destination in space.

SpaceRyde's Solution

AFFORDABLE
Lowest cost per kg of payload of any launch provider

DEPENDABLE
Guaranteed launch window that fits customer schedules

ADAPTABLE
Tailored launch for customer priority payloads

SCALABLE
Launch from anywhere, no launch pad needed

SUSTAINABLE
Zero emissions within the lower layers of the atmosphere

Haghighat, who has a doctorate in aerospace engineering, and Safari, a chemist with an MBA, launched SpaceRyde in 2018. "The first on-demand taxi to space" was chosen as the company tagline.

Haghighat had used Ansys software to solve many engineering challenges in the past, so knew he could rely on it again for this new venture. He contacted Ansys and learned about the Ansys Startup Program, which offers simulation software to startups at reduced prices. Soon, SpaceRyde was a member of the program, which has proven to be invaluable to the company.

"Launching rockets from balloons is not a proven technology right now for launching satellites, so we are playing a little bit of a catch-up to get to market," Safari says, noting that they are in competition with more established commercial space companies. "So, we have to be fast in our development, and using the right software — such as Ansys — is helping us make our development faster or, in some cases, more accurate, precise, and correct."

SpaceRyde's goal is to offer on-demand access to space. Payloads will consist of large batches of small satellites, each weighing less than 150 kilograms, for communications, Earth observation, and remote sensing. The first step is using a balloon to lift a rocket resting horizontally on a carrier beneath the balloon to a height of 30 kilometers. Once the rocket carrier has stabilized and pointed the rocket in the desired direction, the rocket is dropped from the carrier and ignited to launch the payload farther into space.

Launching payloads from stationary launch pads on Earth is expensive due to the amount of fuel burned to escape the force of gravity and the effects of the atmosphere. Rockets use less fuel when launched in vacuum conditions above 99% of the atmosphere. SpaceRyde's $1M rocket is cheaper than any other rocket available today, according to Safari.

Stationary launch pads also have costly maintenance and refurbishments between launches, and place severe restrictions on where launches can originate. A launch may have to be delayed if a major storm converges on a launch pad in Florida. With a balloon-based system, there is no permanent hardware to maintain, and the launch site can be moved outside of a storm's range to avoid any delays.

"If you have different places that you can launch from, then you can make rocket launches as frequent as airplanes fly these days," Safari says. Cheap rockets and flexibility of launching from anywhere is how SpaceRyde makes on-demand access to space possible.

“In 2022, we’re going to launch into space with our full-scale rocket and rocket carrier, joining the big leagues of the handful of other rocket companies that have been able to do that.”

MANAGING COSTS AND SAFETY USING STRUCTURAL SIMULATION

To keep costs down, SpaceRyde resorts to simulation tools and performs lots of virtual tests. Among their tools, Ansys Mechanical stands out.

To virtually test structural components, SpaceRyde engineers used Ansys Mechanical to ensure the strength of different pieces of the launch vehicle as well as the ground equipment, including the critical load-bearing parts in the rocket engine test stand.

The intuitive contact tools in Mechanical enabled the high-fidelity modeling of pin joints, bearings, and large jointed assemblies under static and transient loads without having to manually input every contact region. Shared topology support allowed SpaceRyde to mix shell and solid elements in their models to obtain efficient analyses of complex weldments that combine thin and thick structural members.

Ansys Mechanical was also used to analyze the stress on actuators mounted to the rocket’s body. These actuators push on the bell section of the rocket nozzle to control the orientation of the rocket, which can induce considerable amounts of stress in the nozzle walls. The results of these analyses revealed how the design can be modified to improve numerous characteristics such as weight, manufacturability, and cost without compromising structural integrity.

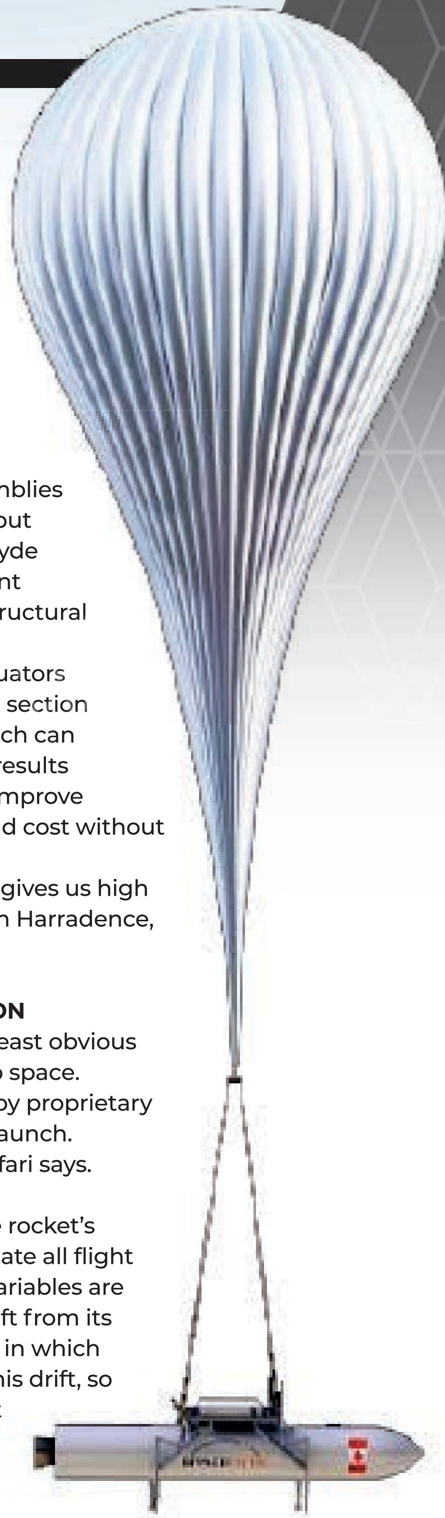
“Through modeling of complete assemblies, Ansys software gives us high confidence in the design of safety-critical projects,” says William Harradence, who is part of the propulsion engineering team at SpaceRyde.

LAUNCHING ROCKETS FROM BALLOONS REQUIRES PRECISION

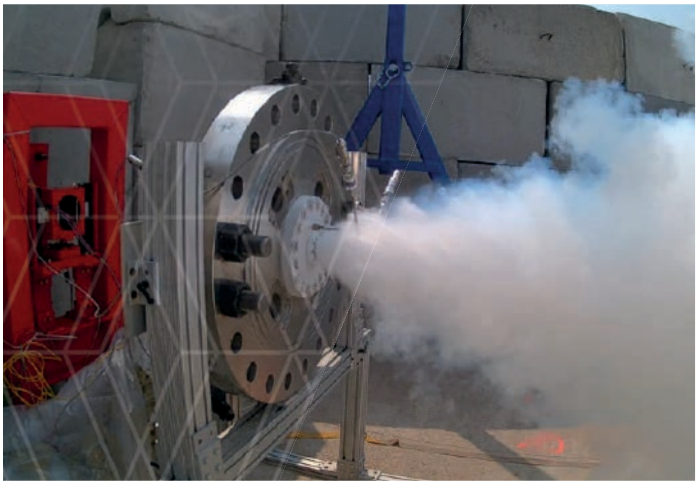
The most innovative component of SpaceRyde’s system is the least obvious one at first glance — the rocket carrier that the balloon lifts into space. Surprisingly, the carrier includes an array of sensors controlled by proprietary software to aid in the precise positioning of the rocket prior to launch.

“The first year was devoted to work on the rocket carrier,” Safari says. “I call it the heart of our operation.”

When launching from a stationary launchpad, you know the rocket’s exact starting coordinates and its destination, so you can calculate all flight parameters beforehand using powerful supercomputers. But variables are introduced when launching from a balloon. The balloon can drift from its predicted launch point. The rocket trajectory and the direction in which the rocket has to be pointed needs to be updated because of this drift, so GPS and other sensors on the rocket carrier determine its exact location. Calculations must be done on the fly to determine the optimal flight path that will get the rocket to its desired spot using the limited amount of stored fuel. Turning the rocket carrier so the rocket is pointing in the right direction at launch also requires proprietary technology.



Launching rockets above the atmosphere saves fuel.



Injector test

The reusable rocket carrier becomes extremely important if the launch has to be aborted. For a ground-based launch, you simply stop the countdown and interrupt the ignition process while the rocket is still standing safely on the ground in its launch tower. If a space-based launch is aborted, the launch team must be able to direct the rocket carrier back to a designated spot on Earth safely — with a fully fueled rocket aboard — without causing it to explode on landing. SpaceRyde proved they could do this during a test.

On June 9, 2019, in North Bay, Ontario, they launched their first test flight. Using a smaller test version of the system, they safely landed the rocket on its carrier in the woods with no damage to the equipment. It was ready for reuse as planned.

FLUID AND THERMAL SIMULATIONS FOR BALLOON-BASED ROCKET LAUNCHES

SpaceRyde's engineers have put the Ansys fluid and thermal solvers they received through the Ansys Startup Program to good use, solving other challenges they faced in designing the balloon, rocket, and carrier system.

Simulating Flow

SpaceRyde engineers used Ansys Fluent to predict the shape and quality of the wake below the balloon during ascent. This provides important information regarding air flow around the rocket carrier during ascent and helps estimate the impact of flow on thermal loads, the rocket carrier's control capabilities, and more.

Flow simulation helped SpaceRyde design their rocket engine injector. The engineers used Fluent to estimate the pressure drop across the injector, which enabled them to target a precise design to ensure safe engine operation before manufacturing the injector. Subsequent physical experiments with the machined injector showed nearly identical behavior to that predicted by Fluent.

Thermal Analysis

Using Ansys Icepak, SpaceRyde engineers modeled the thermal environment of the rocket carrier on its journey from the ground to the upper atmosphere. This enabled them to analyze the thermal environment while incorporating complex factors such as the changing solar environment and changing air pressure at elevation. Icepak helped them to evaluate various mission profiles and to accommodate the changing thermal environments.

FUTURE PLANS FOR BALLOON-LAUNCHED SATELLITES

SpaceRyde has grown from a husband-and-wife operation to a company of over 20 people, the majority of whom are engineers. They have received grants from the Canadian Space Agency (CSA) and from the National Research Council of Canada to fund their research and development. They plan to start full commercial operations in 2023. That means they have a lot of work to do before then.

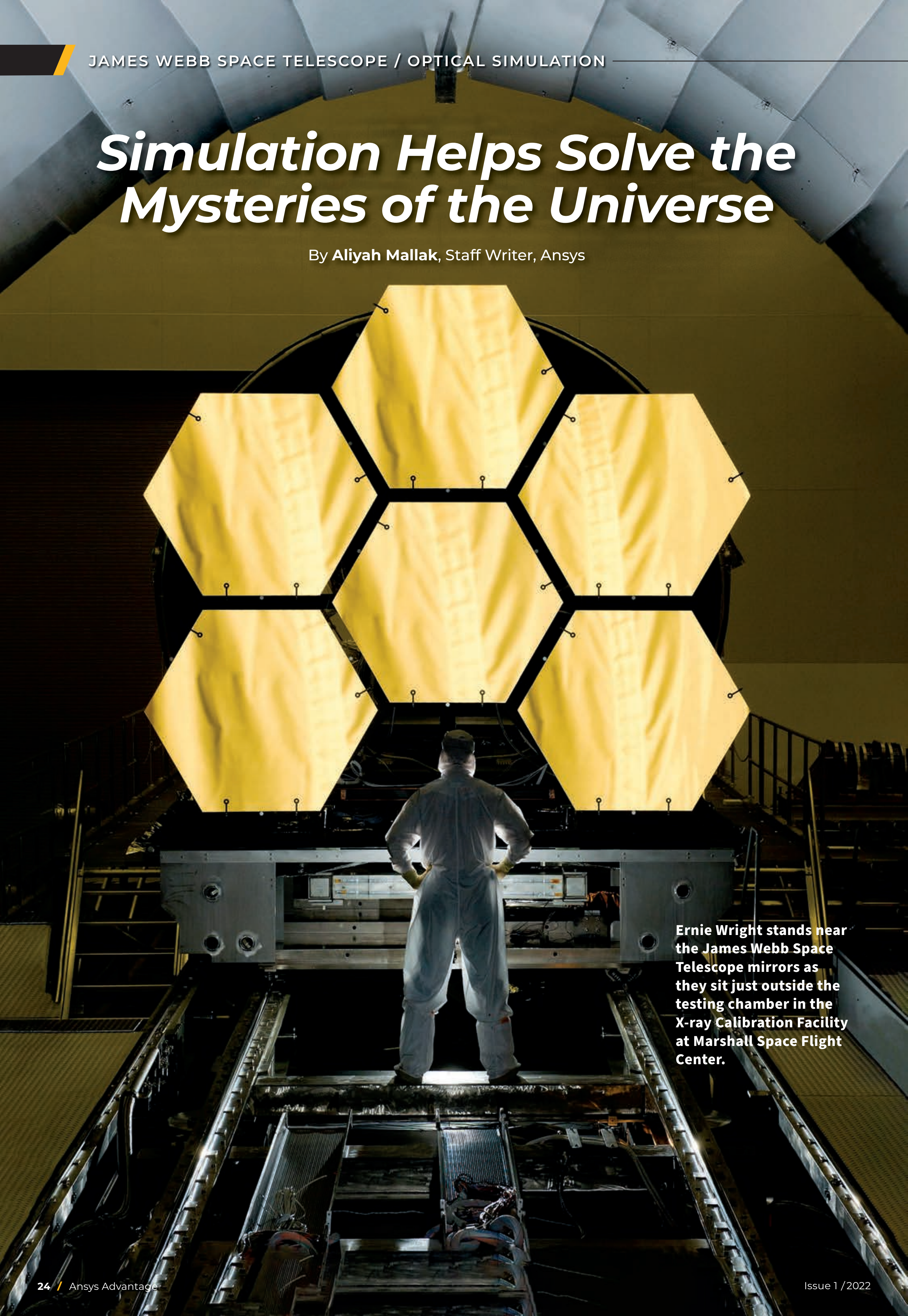
"In 2022, we're going to launch into space with our full-scale rocket and rocket carrier, joining the big leagues of the handful of other rocket companies that have been able to do that," Safari says.

The Ansys Startup Program team will be cheering them on.

For more information about the Ansys Startup Program, visit ansys.com/startup-program.

Simulation Helps Solve the Mysteries of the Universe

By **Aliyah Mallak**, Staff Writer, Ansys

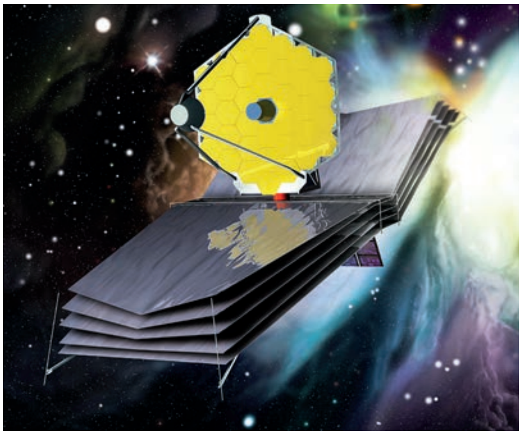


Ernie Wright stands near the James Webb Space Telescope mirrors as they sit just outside the testing chamber in the X-ray Calibration Facility at Marshall Space Flight Center.

With the successful launch of the James Webb Space Telescope (JWST) on Dec. 25, 2021, NASA embarked on a mission to help solve mysteries in our solar system and look beyond to distant worlds around other stars. For decades, the NASA Hubble Space Telescope has provided awe-inspiring images of distant galaxies, nebulae, and stars. However, the JWST is now expected to provide an extraordinary leap in our understanding of the universe. With a primary mirror capable of collecting six times more light than Hubble and longer image wavelengths, JWST will enable scientists to look back in time at the earliest formations of the universe.

JAMES WEBB SPACE TELESCOPE DESIGN

JWST requires pioneering technology not only to get into space, but also to see what has never been seen before. The telescope has the largest and most precise optical instrument ever built, one that is designed to stay extremely cold — nearly minus 400 degrees F, or minus 240 degrees C.



Artist rendering of the James Webb Space Telescope
Courtesy of NASA, ESA, and Northrop Grumman

Engineers had to create a structure that can:

- Operate in perpetual solar radiation
- Fold a structure the size of a tennis court into a rocket
- Fly more than a million miles into space
- Have dozens of mechanisms remotely unfold in an intricate, mechanically orchestrated ballet

Simple, right?

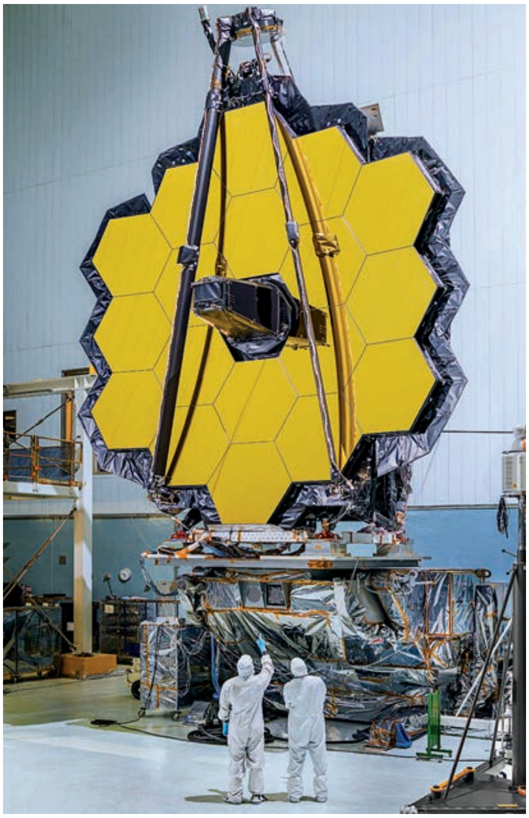
Had any of these delicate maneuvers failed, the mission would have been a loss. With 344 "single-point failures," JWST was one of the riskiest missions in history.

MITIGATING RISK WITH PERVASIVE ENGINEERING SIMULATION

Because the JWST is entirely too large to test everything in person, and testing on Earth isn't the same as testing in space, engineers simulated how the telescope will act in its operational environment. Northrop Grumman is the prime contractor that designed and built the JWST, and simulation played a critical role in that process. "You design it, you build it, you design a computer model to imitate it," says Scott Willoughby, Vice President of the James Webb Space Telescope at Northrop Grumman Space Systems. As the world's leading provider of engineering simulation tools, Ansys is proud to have supported this mission.

DRM, ORBIT DETERMINATION, AND SRP

Engineers used Systems Tool Kit's (STK) Astrogator capability from AGI, an Ansys company, to build complex design reference missions (DRMs). These account for the complex gravitational perturbations in libration point (L2) orbits — which it successfully reached — to estimate station-keeping requirements. They will also use AGI's Orbit Determination Tool Kit (ODTK) to perform operational orbit determinations. To model the pressure of the light from the sun on the large sunshield deployed on JWST, they used the custom solar radiation pressure (SRP) plug-in point in ODTK to insert a proprietary model into ODTK's advanced estimation algorithms.



The primary mirror of NASA's James Webb Space Telescope, consisting of 18 hexagonal mirrors, looks like a giant puzzle piece standing in the massive cleanroom of NASA's Goddard Space Flight Center in Greenbelt, Maryland.

MIRROR POINTING

Engineers used Ansys Mechanical simulations to identify solutions to the precise mirror pointing given the natural vibrational frequencies of the structures. Ansys Mechanical helped determine the effects of having a connected, segmented mirror. The segments respond to disturbances the same way a monolithic mirror would.

MIRROR ALIGNMENT

Ansys Zemax software simulated the complex optics of the numerous individual gold-plated mirror segments in the telescope. The 18 hexagonal segments of JWST's primary mirror function as a single mirror, with a surface smoother than 100 nanometers. Engineers used Zemax software to design and test every step of the alignment process, from the initial segment search to the final fine phasing. (See "Shooting for the Stars with Space Telescope Development" on the opposite page.)

They designed a unique "coarse phasing" step that analyzes spectra to correct piston errors — a segment that is too far forward or behind the others — between the segments. Zemax models were also used to design the multifield step, the final check of the alignment. By building a physical test bed one-seventh the size of the telescope, engineers used Zemax to simulate each alignment step before running the alignment in real hardware. Flight models then translated the test bed telescope to the real alignment situation that occurred on orbit. Flight models in Zemax generated statistical models that could predict the most likely state of the primary mirror at each step in the alignment process to support the design of the segment actuators and predict when other onboard instruments will receive useful information.

Due to the size of JWST, gravity when testing on the ground, and JWST's passive cooling system, simulations of all kinds were stretched to the limits of their accuracy during the development of the telescope. Ground-based testing verified pieces of the simulations that could be directly tested, and the verified models then predicted the on-orbit behavior of the observatory. Data from the observatory will perform final verifications of many of the models. Ansys software users are designing the technology of the future, and Ansys wishes the JWST team the best of luck in their amazing mission to understand the universe.

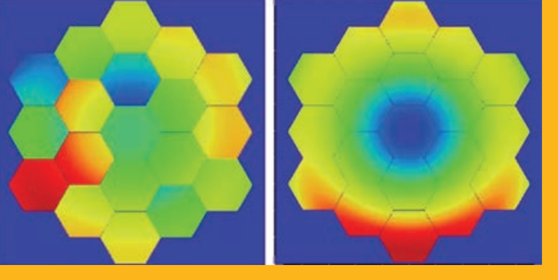
Engineering teams at NASA's James Webb Space Telescope Mission Operations Center at the Space Telescope Science Institute in Baltimore monitor progress as the observatory's second primary mirror wing rotates into position.

WOMEN IN TECHNOLOGY:
SHOOTING FOR THE STARS WITH SPACE TELESCOPE DEVELOPMENT

The development and creation of the James Webb Space Telescope (JWST) involved many bright and talented engineers. Erin Elliott, an optical engineer at Zemax, was one of them. Erin majored in physics and astronomy at the University of Minnesota and received her doctorate in optical science from the University of Arizona. She was hired by an optical engineering firm right out of college, where she worked on telescope designs and proposals, Earth-imaging satellites, and the JWST.

ALIGNING THE MIRRORS WITH ANSYS ZEMAX

Erin was a key part of the team that designed and built the hardware and software used to align the 18 hexagonal segments in the JWST's primary mirror. The segments must be aligned to better than 100 nanometers using imagery from the near-infrared camera (NIRCam) on the telescope. To prove that the primary mirror of JWST could be aligned with such precision, the team used Ansys Zemax software extensively. Erin also created custom add-ons for Zemax that analyzed the aberration signals produced during the multifield step.



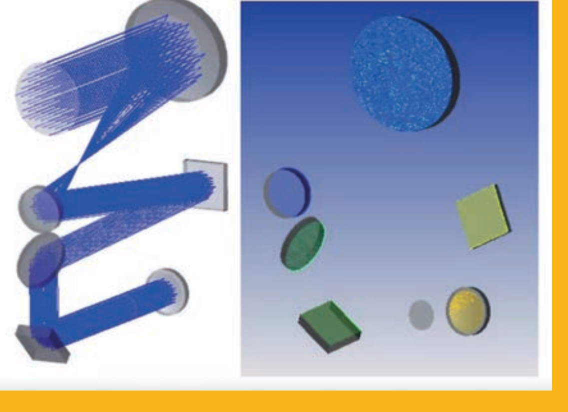
An optical model of JWST-like primary mirror segments, with piston errors (left) and in its perfectly aligned state (right)

"With these custom add-ons, we learned that a combination of aberrations could be used to check the final alignment," Erin explained.

Erin led the effort to design the alignment hardware for the NIRCam and created custom software to statistically analyze the state of the flight telescope at each point in the alignment process. Using this software, the team translated results from the test bed telescope to the on-orbit telescope using models of the full JWST system.

CHARTING A NEW COURSE

Following her work on JWST, Erin changed course by joining Zemax's optical engineering software team. As part of the R&D team at Zemax, she developed STAR software, which adds analysis of stress and thermal perturbations to optical models in Zemax OpticStudio. She designed and prototyped multiple tools for STAR and worked to make the software highly visual and easy to use. She currently leads the Applications Development team, which provides technical leads for new features in Ansys Zemax software products, ensuring that they meet customers' needs.



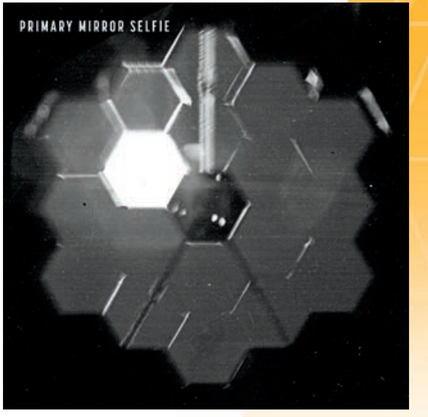
Using the Ansys Zemax OpticStudio STAR software to add deformations (generated by finite element analysis) to the optical model of a mirror system

SUCCESSFUL DEPLOYMENTS TO DATE

As of this writing in early March 2022, the JWST deployments have all gone according to plan. Highlights include:

- **JANUARY 4, 2022:** The JWST team fully deployed the spacecraft's 70-foot, five-layered sunshield, which provides a sun protection factor (SPF) of about one million, to protect the telescope from the light and heat of the sun. With the dimensions of a full-size tennis court, the sunshield was folded inside an Arianespace Ariane 5 rocket's nose cone prior to launch.
- **JANUARY 8, 2022:** The team fully deployed the 21-foot, 18-segment, gold-coated primary mirror. Like the sunshield, the JWST's primary mirror had been folded into the nose cone of the rocket in two wings of nine mirrors each. Unfolding the hexagonal segments of the primary mirror took two days to complete.
- **JANUARY 24, 2022:** Ground teams fired JWST's thrusters to maneuver it around the sun at its intended orbit, the second Lagrange point, or L2, nearly one million miles from Earth.
- **FEBRUARY 3, 2022:** Alignment of the telescope's mirrors began, the first step in a three-month-long process.
- **FEBRUARY 13, 2022:** The JWST took its first unfocused photos of a single star. Because the mirrors were not aligned yet, the single star appeared as 18 separate spots of light on the image. It should come as no surprise in this era of selfies that the JWST then took a selfie of its mirrors — not out of vanity, but to help with their alignment.

The JWST team is currently still aligning the mirrors and waiting for the telescope to cool to its operating temperature to look at the earliest stars and galaxies in the universe from the heat they emit using JWST's near-infrared camera (NIRCam).



"Selfie" taken by JWST of its primary mirror, captured by a special lens within its near-infrared camera (NIRCam)

The JWST's first blurry image of a single star, appearing 18 times for each of the 18 unaligned mirrors. When the mirrors are aligned, the star will appear as a single sharp image.

Tackling Challenges in the NFL With Simulation

By **Jennifer Procario**, Staff Writer, Ansys Advantage

The KOLLIDE football helmet features an innovative system that uses 95 pads to enhance shock absorption and better withstand impact.

With an increasing number of concussions among players in the National Football League (NFL) and a growing movement to prevent head injuries, the organization launched the NFL Helmet Challenge to solicit designs for helmets that are better equipped to sustain impact from varying angles. Rising to the task, a Canadian-based group of innovators formed KOLLIDE to meet this challenge head-on by using Ansys' industry-leading simulation solutions, with the biggest assist from Ansys LS-DYNA.

The trusted explicit solver did not disappoint. KOLLIDE secured a spot as one of the NFL's top three awardees for their helmet's innovative and protective system, which employs 95 pads and an underlying supportive, 3D-printed matrix structure. With $550,000 in grant funding to advance their prototype, KOLLIDE is headed into overtime with plans to further upgrade the helmet and scale the manufacturing process.

THE KICKOFF

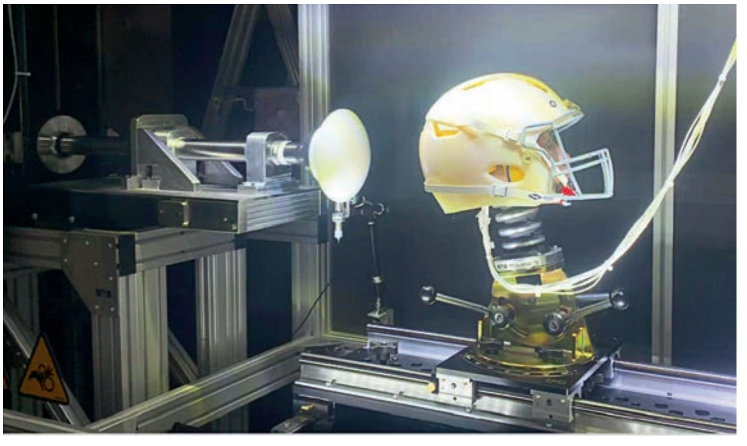
Intrigued by the challenge, Kupol, an advanced manufacturing company, gathered fellow local innovators from startups and niche companies to work together toward a solution in collaboration with the École de technologie supérieure (ÉTS), a decades-old engineering school in Montreal. Together with Kupol, KOLLIDE's lineup includes industrial design firm Tactix, simulation company Numalogics, and software firm Shapeshift 3D.

KOLLIDE took up the challenge in November 2020 with a one-year deadline. However, 12 people could not meet in person for 10 months due to the pandemic, which added another challenge in itself. Given the in-person meeting limitations coupled with budget restrictions for prototyping, the team went digital, turning to virtual design and testing. Implementing simulation as a collaboration tool not only united the team despite member location, but it was also critical to saving development time and costs.

Additionally, the NFL provided four open-source finite element (FE) models of helmets with associated impact test methods and equipment.

"The bench test for testing the helmets is an expensive piece of equipment," says Franck Le Naveaux, a research coordinator for KOLLIDE at Numalogics. "Testing virtually with Ansys LS-DYNA is helpful to improve our speed and reduce costs related to prototyping."

For nearly 10 years, Numalogics has worked with Ansys and the SimuTech Group, an Ansys Elite Channel Partner. Until now, Numalogics experts relied mainly on implicit modeling to simulate medical devices and their interaction with the human body; however, to simulate helmet impacts, explicit modeling was required. Nine meters per second — the highest impact speed in the NFL's helmet testing protocol — may be slower than typical simulations that use LS-DYNA, but it's much faster than what a surgeon does with his hands. Prior to the challenge, Numalogics experts had the most experience using Ansys Workbench for more static, or structural, simulations. But with the speed of impact and composition of the prototype itself, they knew there was only one explicit solver for the job.



Ansys LS-DYNA and 3D printing reduced physical prototyping costs and accelerated product development.

"Materials used in the helmet are fairly complex," says David Benoit, Biomechanical Simulation R&D Specialist at Numalogics. "Being able to simulate materials that are highly nonlinear, viscoelastic, and strain-rate dependent was a challenge. Ansys LS-DYNA already had all of those profiles."

MAKING A PLAY WITH SIMULATION

A different modeling approach was needed to find the optimal trade-off between model accuracy and solving time to iterate quickly. The team strategized how to represent the lattice structures best, trying to understand how to model each wall of the structures. Although the predictive results of the simulation by faithfully representing the lattice microstructure of the pads were promising, too many computing resources were required to simulate the whole helmet. As a solution, they homogenized the lattice models to use fewer computing resources. Still, with 400,000 elements to analyze at the start, KOLLIDE's simulation journey was certainly not express. Ultimately, each type of simulated impact ran in about three hours on 16 cores. The simulations were performed at three different speeds to demonstrate diverse impacts.

KOLLIDE approached the helmet as a system comprising two main parts: the liner and the shell, setting the facemask aside for last. And simulation played a role in each part of the system, allowing the team to test multiple types of liners and shell features while mitigating risks of late design failure.

"Because the helmet is a system, you cannot optimize a single component without affecting the behavior of the whole helmet," says Benoit. "I don't see us being able to provide a fully optimized helmet for the challenge without simulation."

“Materials used in the helmet are fairly complex. Being able to simulate materials that are highly nonlinear, viscoelastic, and strain-rate dependent was a challenge. Ansys LS-DYNA already had all of those profiles.”

The KOLLIDE Helmet comprises 95 shock-absorbent pads.

One of the KOLLIDE helmet’s most essential features is its unique liner composed of networks of lattice-filled, 3D-printed structures. These networks absorb a large amount of energy and enable the pads to have flexibility and movement relative to each other, which boosts shock absorption depending on the direction of impact. Additionally, the group experimented with different printing techniques and materials to consider how lattice structures behave and sustain multiple types of impacts at different temperatures.

The final pads were printed using fused deposition modeling (FDM). For the shell, KOLLIDE tested different compounds to find the best material for sustaining impact before opting for a soft shell to better distribute impact onto the liner’s inner padding. The rigidity of the pads and shell were calibrated using Ansys LS-OPT, a design optimization tool built into LS-DYNA, allowing the development of a fully automatized optimization workflow. In another success, the finished pads are able to shear and mitigate rotational acceleration, which has been found to be the most potentially damaging type of acceleration for the brain.

As a last step, the team created the face mask, which was less of a redesign and more of a fit adjustment to the new liner and shell.

MOVING THE CHAINS: WHAT’S NEXT?

KOLLIDE’s final product looks like a regular helmet but holds advanced features that highlight its lattice structure, such as its customizable fit. There are 95 shock-absorbent pads mounted on a sling in suspension with the shell — an area where simulation helped to improve energy distribution.

“This project would not at all have been possible without using simulation, given the fast time frame. We would have submitted a helmet, but not a good one,” says Naveaux. “Also, with designers working with computer-aided design (CAD) software, bringing concepts to us and testing them, simulation was critical to an efficient workflow.”

Though no team met the NFL’s goal, KOLLIDE scored just above the desired rank. The helmet’s promise was so impressive that the team was selected with two other finalists to receive funding to advance their prototypes.

KOLLIDE is gearing up to do just that, focused on exploring new concepts and scaling 3D printing to produce an upgraded helmet at a faster pace to meet the HPS goal. In accordance with the grant, the NFL will follow KOLLIDE’s progress for a year. The teams have six months to submit a new prototype, then another six months to produce an advanced final product.

NASCAR's Next Gen Race Car *Proven Safe by Simulation*

By **Tim Palucka**, Managing Editor, Ansys Advantage

NASCAR's 2018 announcement that it would develop a new type of race car, called Next Gen, which would be used as a platform by all race teams in the NASCAR Cup Series, predictably raised some eyebrows and caused an outcry among race teams and fans of the sport. With all the things that could potentially go wrong in designing a new race car, why take the risk? And why throw away decades of innovations created by engineers, mechanics, and drivers to optimize their car's performance?

“From an engineering standpoint, our confidence in Ansys simulation is high. The results of a physical crash test are not a surprise to us anymore.”

But NASCAR officials had their reasons. Fan attendance at racetracks was down, and the stock cars that raced under the banner of the National Association for Stock Car Auto Racing (NASCAR) now bore little resemblance to the actual stock cars rolling off assembly lines and into dealerships around the world. In fact, though they were touted as the latest in automotive engineering, the race cars were using highly modified versions of automotive technologies of the 1960s and ’70s, tweaked for ever greater performance by engineers over the years.

John Probst, senior vice president of Racing Innovation at NASCAR, sums up the challenge as one of “relevance.”

“We need a racing platform that will make us more relevant to our original equipment manufacturers (OEMs) and our fans,” Probst says. “One thing that was plaguing us was that the suspension technology on the car was from the 1960s. We hadn’t really migrated with the pace of the automotive industry very well. With respect to that, our race car bodies were more like sedans than racing coupes. And it’s no secret that the internal combustion engine has a lot of pressure on it from electric hybrid and fully electric powertrains. So, we had to become more relevant to our fans, our automotive OEMs, our stock car heritage, and the environmental realities.”

On the business side of racing, the cost of owning a race car team was reaching outrageous levels — in the eight figures range — just to start with a modest team of a couple race cars, a driver, and some engineers and mechanics. This was keeping new owners from entering the sport at a time when NASCAR was interested in expanding. The plan for Next Gen was for all teams to order standard parts from the same vendors to build their cars so costs could be reduced and monitored, perhaps opening the doors to a new crop of team owners.

The Next Gen project started with a clean sheet of paper and input from NASCAR owners, teams, and drivers, all of whom weighed in on the front end and identified what they wanted the new car to be able to do. How did they want it to race, and how much did they want it to cost? All their responses were shared with the designers to turn that sheet of paper into a working race car. With plans to launch the Next Gen race cars at the start of the 2020 season, NASCAR had a lot of engineering to do in a short time. Ansys LS-DYNA, a standard NASCAR simulation solution, was always going to play some role in simulating crash tests to reduce the high costs (approximately $500,000 per test) of running physical crash tests. Then 2020 came and the world stood still. Even the crash test facilities that NASCAR had traditionally used closed their doors because of the COVID-19 pandemic. LS-DYNA was needed more than ever.

SAFETY FIRST

“Safety is our number-one priority at NASCAR,” Probst says. “When we feel like there’s a safety issue, we fix the issue and then go find the funds to pay for it. I won’t say money is no object, but we attack safety issues very seriously with whatever resources we need to get to the correct result.”

The NASCAR Next Gen racecar features a new sequential five-speed shifter.

Ensuring the safety of the Next Gen race car fell to John Patalak, Managing Director, Safety Engineering at NASCAR. Patalak became a convert to LS-DYNA simulation while working on a project with a major automobile manufacturer to improve the crash safety of NASCAR

After more than 5,000 Ansys LS-DYNA crash test simulations, it took only two physical crash tests to verify and validate that the design was safe.

drivers. LS-DYNA simulates the response of materials to short periods of severe loading.

"The first time I was really very impressed with LS-DYNA simulation work was in 2011," Patalak says. The project included a simulated human body model to test the restraint system in the automobile, including the seats, the shoulder supports, and the rib supports around the torso of the driver. "Frankly, I was blown away by the level of detail that you could get using LS-DYNA and the human body model — data that crash test dummies simply just don't provide."

While working for NASCAR, Patalak attended graduate school at the Wake Forest University Center for Injury Biomechanics, where LS-DYNA was the standard simulation software for studies of high impact on the human body. He soon became a champion of LS-DYNA at NASCAR.

His expertise and confidence in LS-DYNA were crucial to making progress on the Next Gen car while crash test labs were closed during the pandemic. For the first time, the bulk of the crash testing would be handled by simulation.

PUTTING ANSYS LS-DYNA TO THE TEST

"From the first prototype car level, we committed to meshing out and building a full LS-DYNA car model," Patalak says. "That immediately allowed us to start assessing the car in all the different crash modes that our race cars encounter: frontals, roof crashes, lateral side impacts, rear impacts, and oblique impacts. We then focused our efforts on particular areas where we wanted to change performance or where we saw opportunities to make things better."

Because designs change rapidly in the prototype stage, LS-DYNA was particularly helpful in enabling engineers to make a change and immediately assess it in many different crash modes. For example, the team was trying to get more deformation out of the front clip (the front section of a car's frame, which is designed to crush when it hits another car or a wall) at different points as the project progressed, but they also wanted to ensure that the changes were not unknowingly introducing negative consequences for other crash modes, such as a T-bone.

"With LS-DYNA, we're able to run all crash scenarios and assess the effects of a design change," Patalak says. "Was it holistically a positive change or a negative change, or what were we missing? These are things that you just don't have the resources to do using physical testing. Being able to quantify these effects with simulations really improved the level of confidence we had in the overall design of this new vehicle."

During the design process, Elemance, an engineering company dedicated to human-centered design using the virtual human body model developed by the Global Human Body Models Consortium (GHBMC), performed tests for NASCAR using Ansys LS-OPT, a design optimization and probabilistic analysis package. LS-OPT uses an inverse process of first specifying the performance criteria and then computing the best design according to a formulation.

Thousands of bumper simulations were run using LS-OPT, and soon NASCAR became so satisfied with the results that they committed to having these parts built without any physical crash test data.

"These were not prototype parts but pre-production parts," Patalak says. "When we finally had a chance to physically crash test them later on, the validation and correlation between the physical test and the simulation model was uncanny."

GETTING NEXT GEN TO THE STARTING LINE

Over the course of the Next Gen design project, NASCAR and Elemance ran more than 5,000 Ansys simulations to ensure the new car would be safe in any crash scenario. They started with a third-party cloud solution early in the process, then switched to Ansys Cloud, run on Microsoft Azure in mid-2021 to gain the benefits of having a single source for their simulation software needs. They identified the HC-series Virtual Machine, which features 44 Intel Xeon Platinum 8168 processor cores, as the best hardware configuration both in terms of performance and cost.

One of the biggest advantages of using a single software supplier for simulations, according to Patalak, is that you have only one company to call to solve any problems that arise. There is no other company to blame.

"The really nice thing about running jobs on Ansys Cloud was that it was all in-house," he says. "I had very few issues, but just getting anything up and running on a new platform, it takes some time to learn the settings and which clusters to send jobs to, the memory settings, etc. It was nice because it was never someone else's fault. The turnaround time and the support from the Ansys Customer Excellence (ACE) team was great — it was a one-stop shop for troubleshooting and answers."

After more than 5,000 crash test simulations, it took only two physical crash tests to verify and validate that the design was safe and the results of the LS-DYNA simulations were accurate.

"From an engineering standpoint, our confidence in Ansys simulation is high. The results of a physical crash test are not a surprise to us anymore," Patalak says.

GENTLEMEN, START YOUR ENGINES!

But would the Next Gen car be ready for the first demonstration race at the Los Angeles Memorial Coliseum, a stadium converted into a one-quarter-mile short-track racing venue, on Feb. 6, 2022? Despite the skeptics and the ups and downs of the design process, it was — much to Probst's relief.

"If we didn't have Ansys LS-DYNA, I'm not sure the Next Gen car would have been ready for the 2022 season because of everything being so disrupted by the pandemic," Probst says.

In the end, the coliseum race was successful, and was a proper lead-in to the most anticipated event in the NASCAR season: the Daytona 500, which kicks off every NASCAR Cup Series. Held Feb. 20, it resulted in 23 crash reports from the 40 cars in the race, with no significant injuries to the drivers. The Auto Club Speedway 400, held in Fontana, California, on Feb. 27, was similarly successful.

The Next Gen race cars made their public debut in May 2021 and were first demonstrated in an early February race.

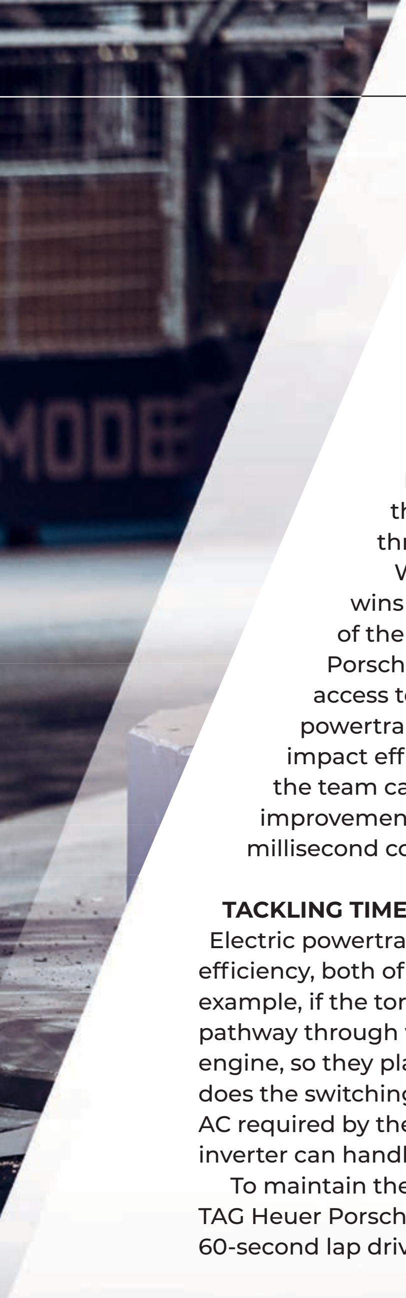
"We were happy leaving Daytona with the knowledge that what we expected to see from our modeling work was playing out on the racetrack," Patalak says. "At Fontana, we had more and different types of crashes and, again, good outcomes. The results are matching very closely with our expectations that were shaped through LS-DYNA, which is showing itself to be true on the racetrack in real crashes."

Probst agrees.

"From a higher level, I would say that we feel like we're off to a phenomenal start with respect to all the targets around the Next Gen project," he says. "There's still a lot of work to do. We're pleased but never satisfied on our end. But when you look at some of our fan metrics, when you ask somebody something as simple as 'Was that a good race?,' those numbers are actually off the charts right now, which we think bodes really well for this car."

TAG Heuer Porsche Formula E Team *Optimizes its Powertrain Dynamics With Ansys Simulation Software*

By **Laura Carter**, Staff Writer, Ansys Advantage



Racing through the streets of some of the most iconic cities in the world, some of the biggest drivers and teams in motorsports compete in the ABB FIA Formula E World Championship. This racing series is designed to push the limits of electric vehicle technology. During these high-speed, single-seater matchups, world-class drivers test their skills over a period of 45 minutes at speeds up to 174 mph (280 km/h) through winding, narrow urban racetracks.

With an ABB Formula E mandate on total energy consumption per team, wins are not just defined by speed, but also by kilowatt hours, so the efficiency of the race car's powertrain is critical to a team's success. For the TAG Heuer Porsche Formula E Team, a win on a race day depends, in part, on the team's access to a simulation environment that can help them quickly analyze electric powertrain component performance and gain insights into the mechanisms that impact efficiency. Using Ansys Maxwell and Ansys Twin Builder simulation software, the team can detect and optimize the smallest of details in the powertrain to realize improvements in an extremely high-efficiency environment where every millisecond counts.

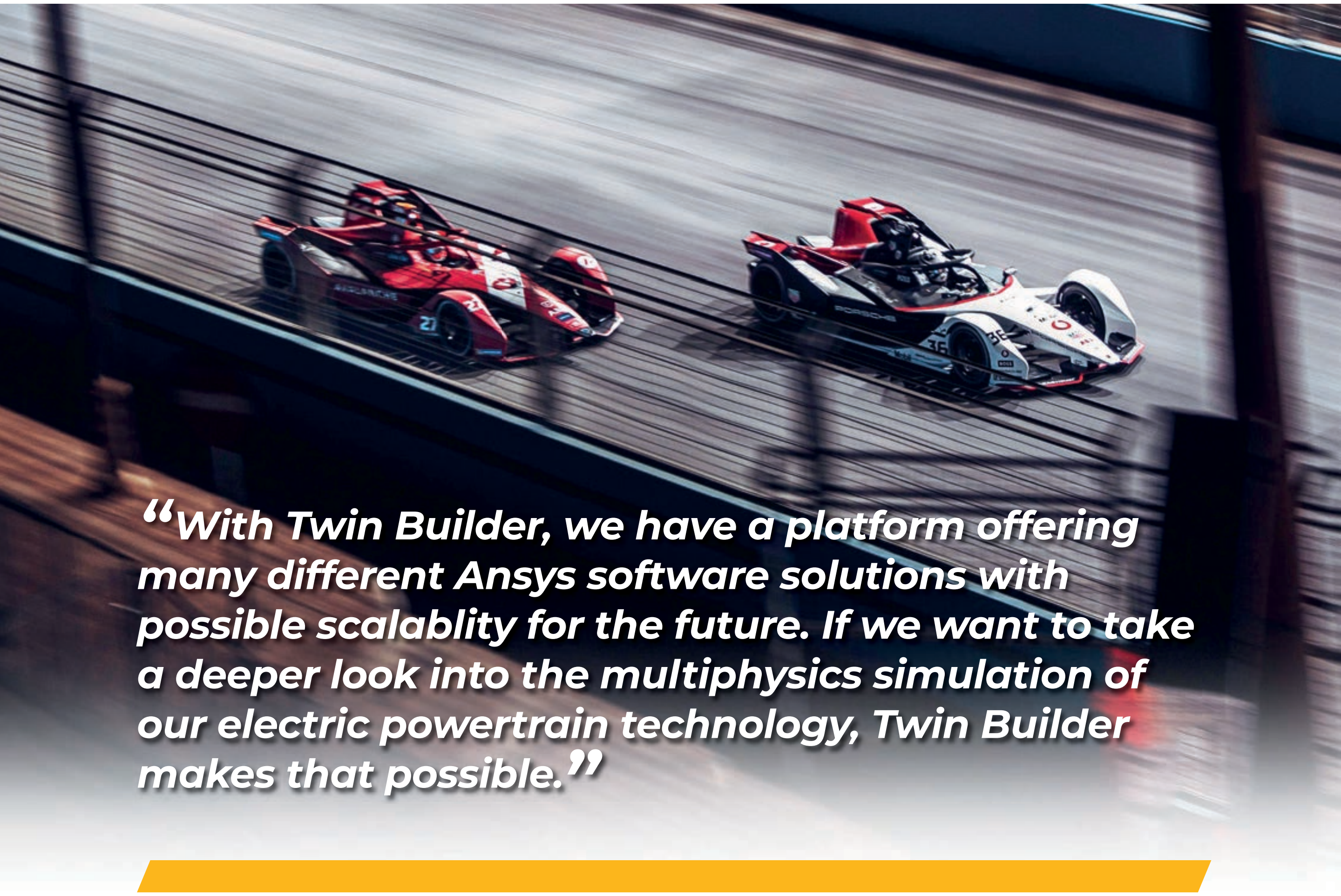
TACKLING TIMESCALE CHALLENGES WITH SIMULATION

Electric powertrain dynamics are ruled by two things: overall efficiency and component efficiency, both of which must be optimized over a wide range of operating points. For example, if the torque or speed is too high or too low, energy is wasted. Inverters are the pathway through which battery power is converted during driving and delivered to the engine, so they play a critical role in overall powertrain efficiency. It is the inverter that does the switching (electrical conversion) from battery DC to high-frequency, high-voltage AC required by the engine during a race. The faster the switch speed, the more power an inverter can handle with fewer electrical losses.

To maintain their competitive edge in the ABB Formula E World Championship, the TAG Heuer Porsche Formula E Team set its sights on tuning their powertrain to achieve 60-second lap drive cycles, which represent the speed of the vehicle over time. Each cycle

INTERACTIVE RACING

What's more exciting than unlocking a power boost for your favorite motorsport? Giving an extra boost to your favorite race car driver in one of the most innovative racing series. Using a simple hashtag on Twitter, a special mobile app, or an official website, you can vote to give the TAG Heuer Porsche Formula E Team a 30 kW "Fanboost" (fanboost.fiaformulae.com) during their next race, big enough to potentially outmaneuver a competitor and light up a car's halo system purple. (Be sure to vote for Porsche Motorsports drivers Andrew Lotterer [99X Electric #36] and Pascale Wehrlein [99X Electric #99] at the next race!) There's also an "Attack Mode" that requires drivers to cross a designated area on the circuit to earn 30 kW of additional power (this time lighting up their car's halo system blue).



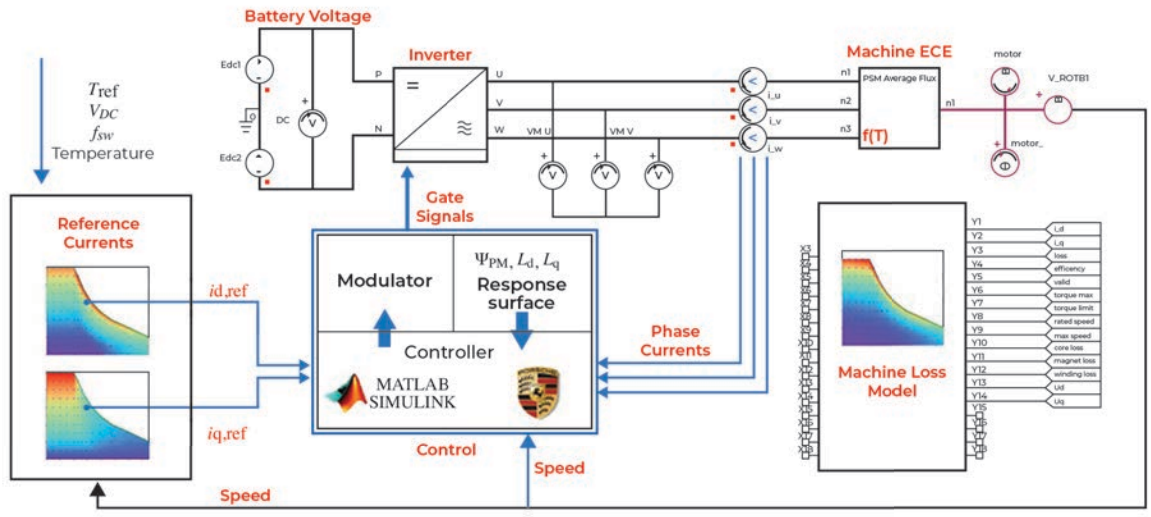
requires one electric period (the duration when energy is delivered) of between 10 milliseconds and 0.5 milliseconds, with an inverter switching period (the rate at which the switching device is turned on and off to deliver this energy) of just 0.1 milliseconds to 0.03 milliseconds. Examining powertrain efficiencies at both the component and the system levels, the team can better address electrical losses to achieve the targeted time scales.

USING ANSYS TWIN BUILDER AND MAXWELL TO UNCOVER ELECTRICAL LOSS

Ansys simulation software plays a big role in optimizing the TAG Heuer Porsche Formula E powertrain dynamics through data collection and analysis of electrical loss. Simulating drive cycles using Maxwell, the team (from the development center in Weissach) can see the impact of voltage changes on battery charge loss to pinpoint and minimize electrical loss during each lap over the course of a race. The team also uses Maxwell separately during one electrical period to simulate the full operating range of the electrical machine and its interaction with the inverter to identify loss results with greater precision. And, faced with the potentially time-consuming process of inverter switch loss calculations, the team adopted Twin Builder to measure switching losses quickly using data from the manufacturer, combined with their own measurements and testing.

With the help of Twin Builder, the TAG Heuer Porsche Formula E Team worked side by side with Ansys to build and use a system-level model to test efficiencies at the component level. The team then implemented and tested specific component attributes of the existing powertrain under a variety of operating conditions, such as the impact of phase currents on inverter switching or induced voltage and torque on machine function. As a result of this careful step-by-step analysis of each component, operating both individually and as part of the entire powertrain, the team from Weissach is now able to estimate individual component performance and electrical loss with much higher accuracy.

POWERTRAIN SYSTEM MODEL IN ANSYS TWIN BUILDER



"Considering that the powertrain was operating at 90% efficiency going into simulation, the accuracy of the simulation software is key," says Martin Füchtner, Technical Project Leader for Formula E at Porsche Motorsport. "The losses, although significant in terms of electrical performance and powertrain optimization, without the help of Twin Builder, are barely discernable."

SUSTAINABILITY FINDS ITS WAY TO THE FINISH LINE

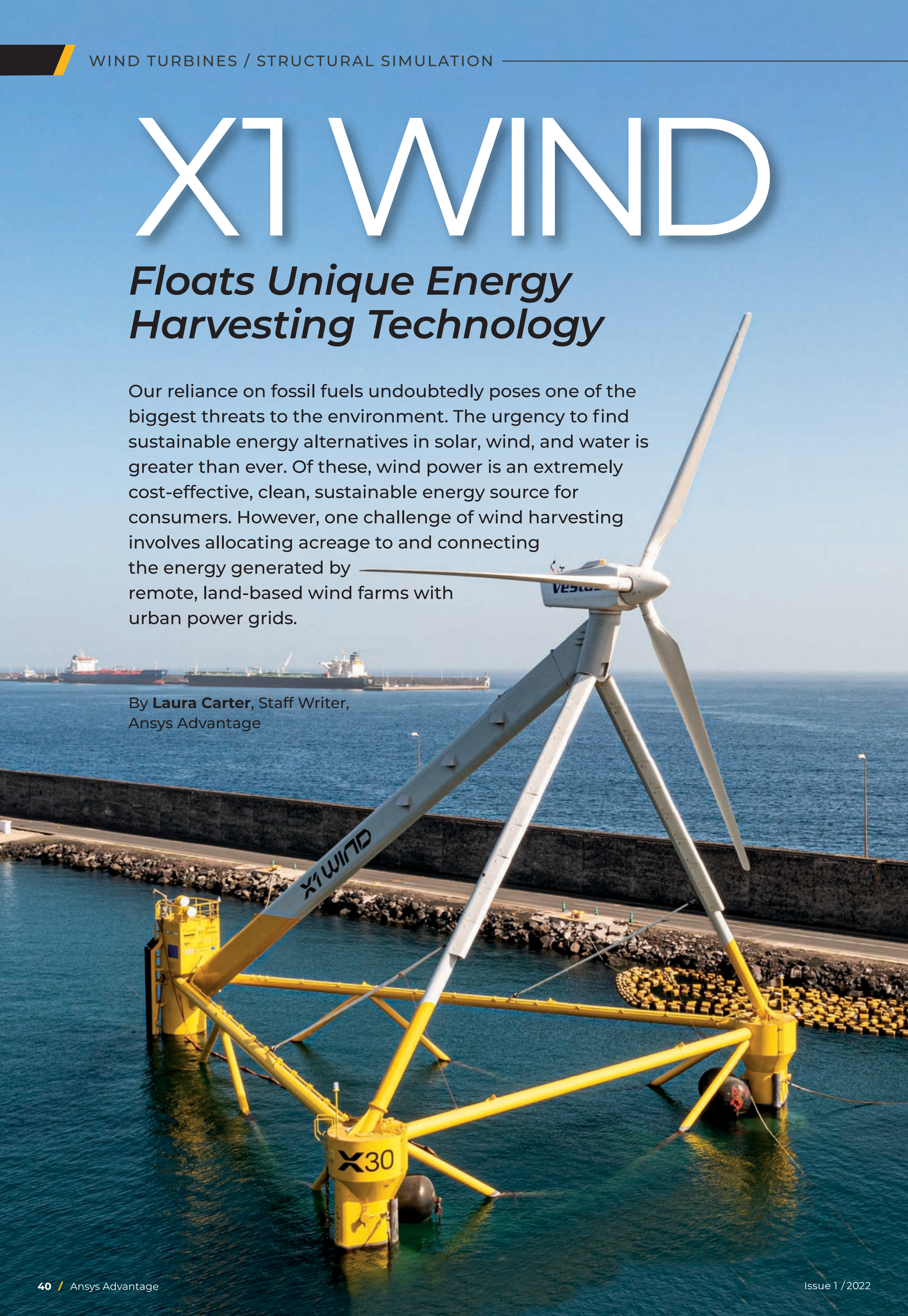
Before hitting the motorsports scene, ABB Formula E began as an idea captured on a napkin at a dinner between FIA President Jean Todt and Spanish businessman and former ABB Formula E CEO Alejandro Agag. Their idea to combine motorsports with sustainability to create a fully electric racing series was initially met with much skepticism. Undeterred by public criticism, they saw a future in e-mobility in racing and the greater contributions they could make in support of a greener, cleaner environment. Both realized their vision in 2014 in Beijing, China, the site of the first ABB Formula E racing event in motorsports, and the rest is history.

In July 2017, the TAG Heuer Porsche Formula E Team announced its intentions to join the first fully electric racing series ever in season six, making their commitment to hit the urban race circuits in 2019. The team has set its sights on winning the Driver and Team championships using Ansys simulation as part of their winning strategy, both on and off the track. Looking ahead, the TAG Heuer Porsche Formula E Team is using Ansys software to optimize the powertrain for a Gen 3 race car, specifically modeling detailed loss mechanisms related to vehicle AC power without increasing simulation duration.

CHARGING INTO THE FUTURE WITH ANSYS

The ABB Formula E World Championship is a fast and demanding test bench for future technologies that requires constant dialogue among development engineers to test new ideas and learn from each other. To meet these demands, Porsche sees Ansys simulation solutions as a good technological foundation for development of its 99X Electric race car, as well as road cars of the future.

"With Twin Builder, we have a platform offering many different Ansys software solutions with possible scalability for the future," says Füchtner. "If we want to take a deeper look into the multiphysics simulation of our electric powertrain technology, Twin Builder makes that possible."



X1 WIND

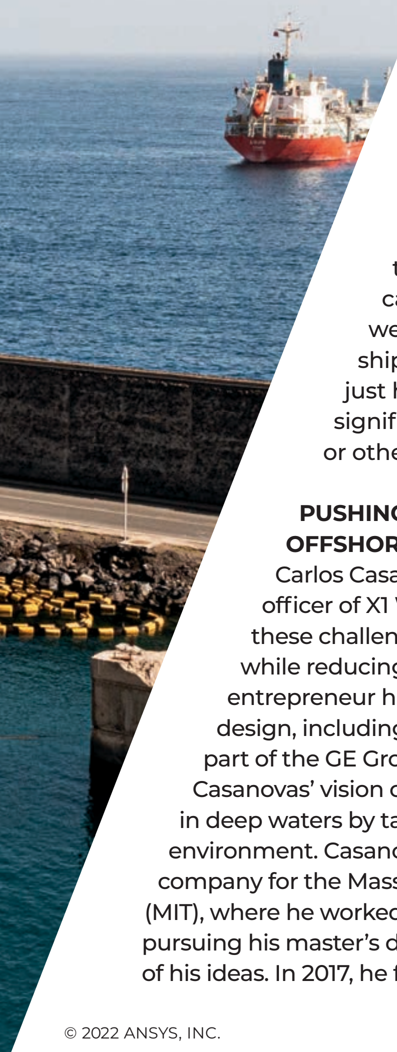
Floats Unique Energy Harvesting Technology

Our reliance on fossil fuels undoubtedly poses one of the biggest threats to the environment. The urgency to find sustainable energy alternatives in solar, wind, and water is greater than ever. Of these, wind power is an extremely cost-effective, clean, sustainable energy source for consumers. However, one challenge of wind harvesting involves allocating acreage to and connecting the energy generated by remote, land-based wind farms with urban power grids.

By **Laura Carter**, Staff Writer, Ansys Advantage

“It’s important to have the right simulation tools so that you can do all the optimization that you need to do, and Ansys checks all the boxes.”

Interior view of the tension leg platform (TLP) floater, to which the mooring lines are connected



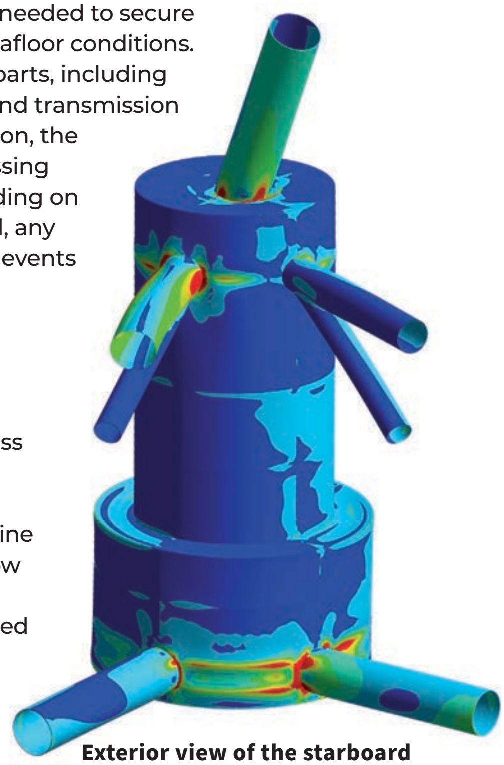
By taking wind energy technology offshore, the prospects for wind power expansion increase dramatically. Our planet’s vast bodies of saltwater offer limitless possibilities for wind harvesting, yet most remain untapped or underutilized due to wind turbine installation costs and maintenance challenges. With the support of the Ansys Startup Program, startup X1 Wind, located in Barcelona, Spain, gained valuable access to Ansys simulation software and developed a disruptive floating wind technology that can mitigate these challenges and radically transform energy production.

Approximately 70% of the Earth’s surface is covered by oceans, representing a 140-million-square-mile area that holds enormous potential for floating wind turbine applications. According to the International Energy Agency (IEA) — an organization dedicated to helping countries achieve sustainable energy objectives through authoritative analysis, data, policy recommendations, and real-world solutions — the technical potential for offshore wind worldwide is 120,000 Gigawatts (GW). This translates into more than 420,000 terawatt hours (TWh) of electricity per year, enough to meet global demand for electricity 11 times over in 2040. So, just how does an offshore floating wind turbine fit into this scenario?

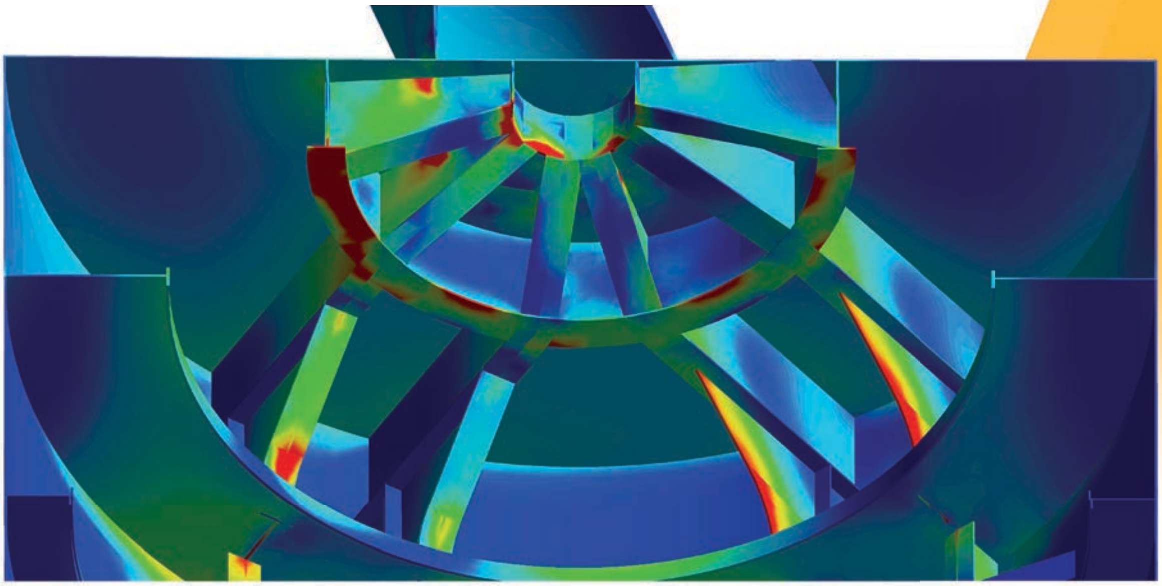
Once moored to the ocean floor, wind turbine networks can be established, connected by high-voltage direct current cables to supply power to coastal mega grids. Of course, there are various logistical challenges that come into play working offshore in deep waters. Initial wind turbine installation is costly and complex, requiring expensive, specialized vessels and the expertise needed to secure a foundation underwater in varying seafloor conditions. A wind turbine also has many moving parts, including the turbine blades, gearbox, generator and transmission cables — some being challenged by erosion, the weather, and occasional damage from passing ships while operating continuously. Depending on just how far offshore the turbines are located, any significant delays due to unforeseen weather events or other logistical challenges can be costly.

PUSHING BOUNDARIES IN FLOATING OFFSHORE WIND DESIGN

Carlos Casanovas, co-founder and chief technology officer of X1 Wind, developed the X30 floater to address these challenges and deliver clean, affordable energy while reducing emissions around the globe. The young entrepreneur has an extensive background in wind turbine design, including time spent working at Alstom Wind, now part of the GE Group. The X30 floater is the realization of Casanovas’ vision of how the wind energy could be harnessed in deep waters by taking advantage of the complex offshore environment. Casanovas would eventually leave the company for the Massachusetts Institute of Technology (MIT), where he worked on his floating design concept while pursuing his master’s degree to further push the boundaries of his ideas. In 2017, he formed X1 Wind to bring these ideas to



Exterior view of the starboard main column with the pontoon termination in an extreme load case



Interior view of the starboard column, showing the internal reinforcements under the main mast connection

life with help from several benefactors, including the Ansys Startup Program. He was first introduced to Ansys simulation software on the job at Alstom Wind, so, for him, it was a natural choice.

"Going into this project, I was already familiar with the software and its capabilities," says Casanovas. "When we started out, we contacted Ansys to purchase our first license and were told we were eligible for their startup program. To get those full-license products for very reasonable cost has been very helpful for our development and testing activities. Plus, many of our senior team members already had experience with Ansys software, so for us it was a no-brainer."

Conventional offshore wind turbine towers are supported by large steel structures anchored to the ocean floor, with the blades affixed to the top of a stationary tower. In deeper waters, installing and maintaining these foundations becomes increasingly impractical. The X30 floater is a self-orienting downwind wind turbine design characterized by a wind turbine in a downwind configuration. Removal of the traditional stationary tower resulted in a unique tripod-like floating structure that is anchored to the ocean floor by a single-point mooring system. The lighter pyramidal design has a lower center of gravity that can engage in more efficient load transmission through tension and compression with low bending moments. The downwind design does not require pre-bent blades like conventional upwind offshore turbines, allowing turbine blades to be lighter, longer, and cheaper.

Casanovas' patented PivotBuoy single-point mooring integrates into the downwind configuration. This unique design enables the platform to passively weathervane, or pivot, in the direction of the

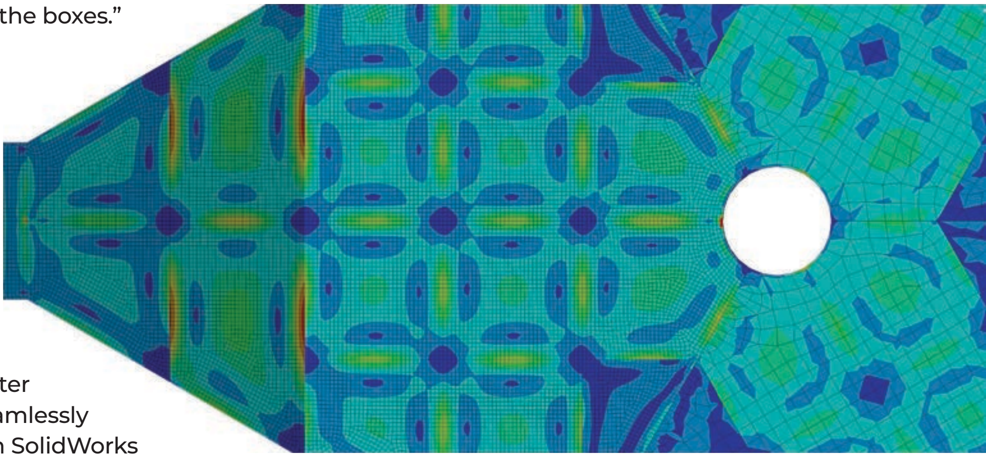
Predicting Future Maintenance with a Digital Twin

Now that X30 floater deployment is in sight, the X1 Wind team is looking to Ansys Twin Builder for predictive maintenance. The X30 system design is packed with sensors, accelerometers, and a few strain gauges collecting and delivering data on system functionality. The full-scale, commercial version of the X30 floater, scaled up by a factor of three, will have many more. Building a digital twin to check and compare structural monitoring through strain gauge data against simulated values, for example, will help the team make predictions regarding how the system will perform over its lifetime. This is valuable information X1 Wind can use to anticipate required maintenance early on and avoid the risk of catastrophic failure.

"When we started out, we contacted Ansys to purchase our first license and were told we were eligible for their startup program. To get those full-license products for very reasonable cost has been very helpful for our development and testing activities."

wind, using mooring lines instead of fixed piles to anchor the system. The single mooring system can be pre-installed while the floating structure can be quickly assembled onshore, then towed using local vessels for quick connection to the single-point mooring system. The result is a more sustainable, reliable energy-harvesting solution that enables modular, scalable construction for quick, cost-effective installation using local infrastructure.

"We need to try to make our floating wind structure reliable and strong enough to remain competitive because, with renewable energy, it's all about the final cost of energy," says Casanovas. "If your floating design is very heavy and expensive to build, you will not be very competitive, so structural integrity is a really fine line for us. It's important to have the right simulation tools so that you can do all the optimization that you need to do, and Ansys checks all the boxes."



Exterior view of the TLP, showing mesh detail

SIMULATING MORE SUSTAINABLE OFFSHORE SOLUTIONS

Ansys simulation software was instrumental in helping X1 Wind meet specific structural requirements for the X30 floating design. It was part of the team's greater workflow, which moved seamlessly at the modeling stage from SolidWorks software to Ansys Mechanical finite element analysis software for structural engineering. Ansys SpaceClaim 3D computer-assisted design modeling software was used as an intermediate step to improve the geometry of the model and optimize it for further analysis. The team then used Mechanical for model analysis of both extreme loads and fatigue to verify that all structural stresses were under acceptable limits. The software was also used to verify the plastic deformation of various components.

The future for X1 Wind is looking very bright. Commissioning their first prototype, X1 Wind plans to install the X30 demo floater somewhere in the waters off the Canary Islands in the upcoming months. Deployment of the PivotBuoy project was funded by the European Commission's Horizon 2020 program, part of a dedicated effort to invest in new solutions to address societal challenges and drive innovation-led, sustainable growth. However, Casanovas was quick to point out that the support of the Ansys Startup Program was extremely valuable in the realization of the X30 floater.

"Considering the requirements surrounding wind energy and the complexities of the architecture as a whole, product development would have been much more difficult without the support of the Ansys Startup Program," says Casanovas. "Without the access to Ansys software needed to help build advanced finite element models, it would just be too much of a challenge for such a small team."

What's Next in Wind Harvesting Technology? **Climbing Cranes, of Course**

By **Laura Carter**, Staff Writer, Ansys Advantage

As the demand for wind power grows, so do wind turbines. When it comes to a wind turbine's ability to harvest energy, size matters. For example, if you build a taller tower, you can take advantage of wind speed increases at higher altitudes to capture more energy. Or, by increasing the rotor diameter and lengthening the turbine blades, you can also increase the area a turbine blade sweeps to increase energy production.

From a distance, it's difficult to comprehend the massive scale of commercial wind turbines. Just how big a scale are we talking about, exactly? To give you an idea, each blade of a Vestas V90-2.0 MW turbine is approximately 144 feet long and weighs approximately 7 tons. That's slightly longer than a Boeing 737 Max 10 commercial aircraft measuring 143 feet 8 inches, and about three times as heavy as an adult rhinoceros at 2.30 tons.

Moving these massive components is not easy. Special high-load transporters and ships are needed, along with mega cranes to assemble them. Installation is an expensive, risky undertaking at extreme altitudes. Unexpected wind gusts can cause a turbine tower to sway or a crane to tip over during installation.

LT1500 view from the top of the turbine (photo illustration)

The LT1500 begins tower assembly (photo illustration)

BUILDING A BETTER CRANE

Liftra is revolutionizing wind turbine installation with their invention of the LT1500 crane, which can "climb" up the wind turbine tower as it is being assembled instead of sitting beside it. After the first section of the tower is in place, the crane grabs onto a flange of that section to lift the next section into place. Then, a "hoist block" is lifted up, attaching to the higher section and pulling the crane up the tower to attach to the higher flange. In a sequence of such grabbing and hoisting moves, an entire wind turbine, including the tower, nacelle, rotor, and blades, can be constructed.

The LT1500 offers manufacturers a more compact turbine installation solution at a considerable cost savings — in man-hours, equipment transport, and installation time — while improving overall safety.

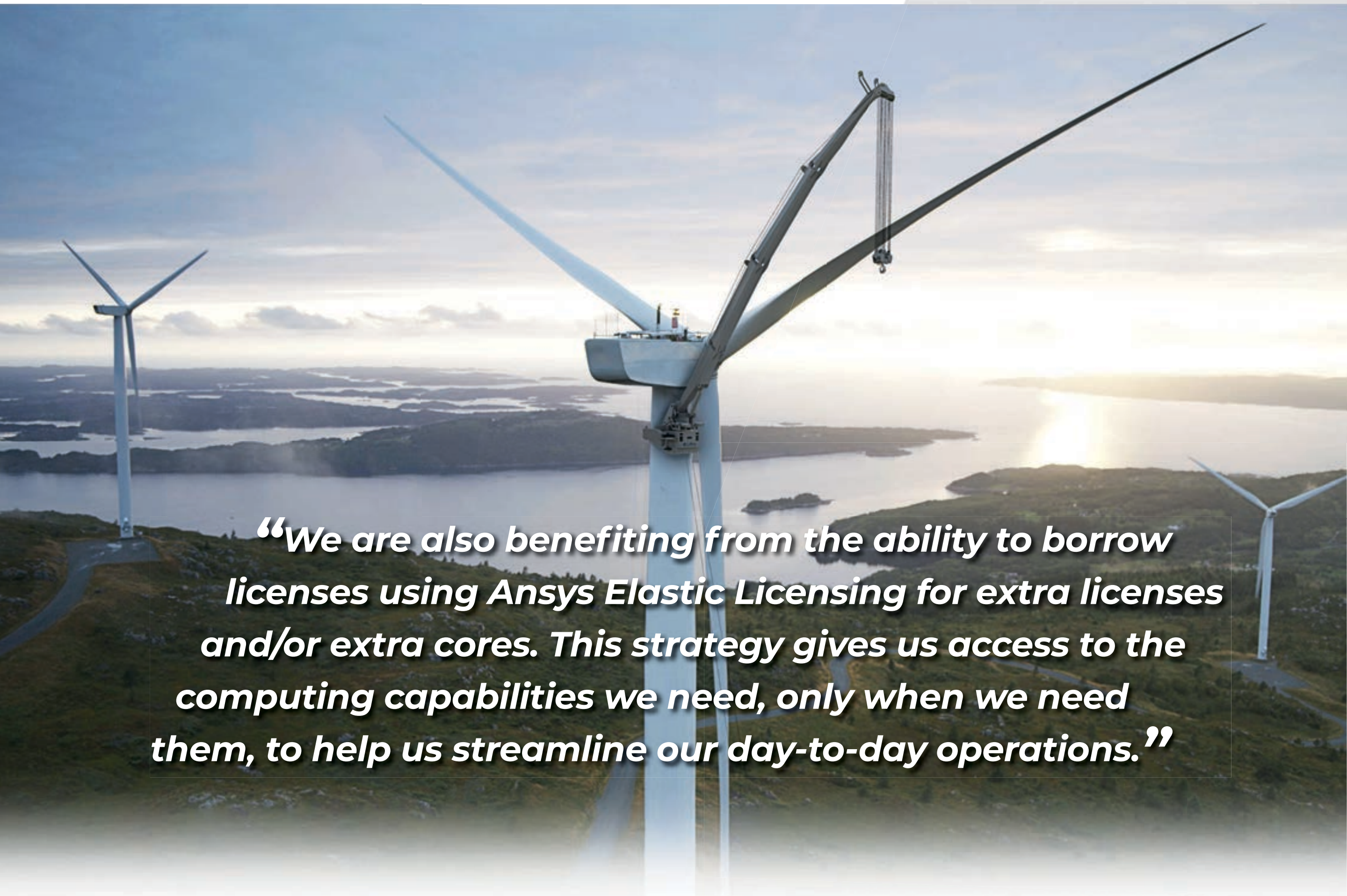
Miguel Adolfo Hoffmann, a senior engineer specialist on the Liftra team, helped review this design effort. Hoffmann serves as Liftra's internal support for Ansys solutions and is frequently consulted to solve complicated calculation questions.

ASSESSING THE LOAD

This unique crane application requires an exceptionally strong interface to work. Of particular concern for Hoffmann was the integrity of the bolted flanges (connecting points of the wind turbine tower) where the crane attaches and their load-bearing ability. Any compromise in design could cause the flanges to bend, risking damage to the tower.

To properly assess these load-bearing characteristics, a team of engineers established four contact points along the tower. Using Ansys Mechanical, the team tested the stress distribution and load distributions among all four points to ensure the load was equally dispersed among them. Verifying the strength of the tower during the mounting of the crane was especially important.

"We rely heavily on Ansys Mechanical to see what happens in the interface between the LT1500



"We are also benefiting from the ability to borrow licenses using Ansys Elastic Licensing for extra licenses and/or extra cores. This strategy gives us access to the computing capabilities we need, only when we need them, to help us streamline our day-to-day operations."

crane and the tower," says Hoffmann. "We are especially interested in checking that the strength of the tower is okay when we mount our crane. We normally run two analyses to make an elastic and elasto-plastic analysis, which is mostly focused on the tower, to see if we have any permanent deformation or not. If we do, we can determine how much deformation we actually have."

Running multiple finite element structural simulations both in Ansys Cloud and on their own workstations, Liftra's team used Mechanical to perform static analysis of the loads involved when the crane is attached to, and climbs up, the wind turbine tower as it is being assembled. Both linear and nonlinear materials models for elastic-plastic analysis were run in the cloud to determine the possibility and extent of any permanent deformation to the tower.

The linear material model was run in Ansys Cloud on an HC Virtual Machine from Microsoft Azure featuring 44 Intel Xeon Platinum 8168 processor cores with a run time of 1 hour, 6 minutes. This model includes 1.8 million nodes with many nonlinear contacts (more than 100 frictional contacts). The nonlinear model was identical to the linear model but used nonlinear material, with a run time of 5 hours, 2 minutes.

The initial runtime results in Ansys Cloud inspired Liftra to run further tests on their own workstations. Testing with a small model (1.2 million nodes, bonded) and a large model (3.2 million nodes, frictional), the team wanted to compare and confirm solving times on a local workstation versus the Ansys Cloud.

These tests showed that the analysis of the larger model was three to five times faster on Ansys Cloud. On average, the team realized four times the speed by running simulations on the cloud. For smaller models, a lower speed increase (4 core to 44 core is only two times faster) did not correlate well with team objectives, so it made more sense to use the cloud exclusively with the larger models.

TAKING A HYBRID APPROACH TO HIGH-PERFORMANCE COMPUTING

Working with Ansys partner EDRMedeso Digital Labs, Liftra relies on Ansys Cloud, their own licenses, and Ansys Elastic Licensing to borrow additional licenses or cores to run static analysis on the LT1500 crane. Taking this hybrid approach to high compute, Liftra can effectively go from 4 to 6 cores on their workstations, resulting in a 46% increase in overall computing power.

Elastic Licensing gives Liftra engineers the flexibility they need to run multiple licenses concurrently. They can easily rent a license for 10 or 15 minutes to get more immediate results, without having to stop mid-analysis to obtain a full license or wait for one to become available. With 11 users in Spain and 36 users in Denmark, Liftra's ability to borrow licenses or use split licenses has become a very cost- and time-effective feature for the team.

LT1500 view from the top of the turbine (photo illustration)

"When running analysis during the day, we will use Ansys Cloud to get fast answers, but when going home, we will run some analysis overnight using our workstations," says Hoffmann. "We are also benefiting from the ability to borrow licenses using Ansys Elastic Licensing for extra licenses and/or extra cores. This strategy gives us access to the computing capabilities we need, only when we need them, to help us streamline our day-to-day operations."

DISCOVERING NEW SOLUTIONS IN WIND HARVESTING TECHNOLOGY

Liftra is recognized by the international wind turbine industry as one of the top specialists in custom lifting and transportation solutions. The company works with manufacturers from around the world to design universal installation and maintenance equipment that's generic enough to be used in most wind turbine applications to solve some of their largest challenges.

The LT1500, Liftra's latest invention, is compact enough to be transported by eight trucks (a conventional crane needs 40–50) and assembled on the ground, cutting assembly time while substantially reducing the overall site footprint. The crane's power pack and the hydraulics for the power pack required for the main hoist, along with the external generator that initially charges the power pack, are all operated from the ground. Using battery power enables the team to recover energy during downward hoists that can be reused when hoisting up the tower.

Once the tower is constructed, numerous components, including the gearbox, the transformer, the hub, and the plates, are mounted with guidance from the team at the top.

REIMAGINING WIND TURBINE INSTALLATION WITH ANSYS

Using Ansys Mechanical simulations, Liftra engineers are reimagining wind turbine installation with a climbing crane design that enables faster wind turbine installations. Having the crane climb the tower as it builds reduces the footprint of the installation site, reducing environmental impacts and improving worker safety on the ground.

For Hoffmann and team, the LT1500 represents another positive step on the journey to net zero emissions. Of course, Ansys — and simulation — will continue to serve as their guideposts along the way to invent something even bigger.

"We have always been using Ansys Mechanical for our static calculations, and we plan to continue using it," says Hoffmann. "We have a lot of design standards to think about. When you derive calculations using Ansys simulation, you can walk into a third-party review with the confidence of knowing the Ansys brand is behind you."

Blade Fatigue Assessment at Bewind Is a Breeze With PyAnsys

By **Daniel Kowollik,**
Lead Engineer, Blade,
Bewind, and
Fabio Pavia,
Senior Product Manager,
Ansys

Wind energy is one of the world's most promising sources of renewable energy. Interest in this emission-free natural resource has grown even more as concerns over air quality and climate change continue to rise globally.

As demand for wind energy increases, the need to advance technology to produce larger, quieter, and less expensive wind turbines also intensifies. As engineers work to make these improvements, they must also ensure the safety of components like the improved blades by verifying their structural integrity and fatigue resistance.

The 28 engineers at Bewind GmbH, a German systems design and engineering firm, have more than 400 total years of wind industry experience. They use PyAnsys to optimize the fatigue assessment of their wind turbine blades.

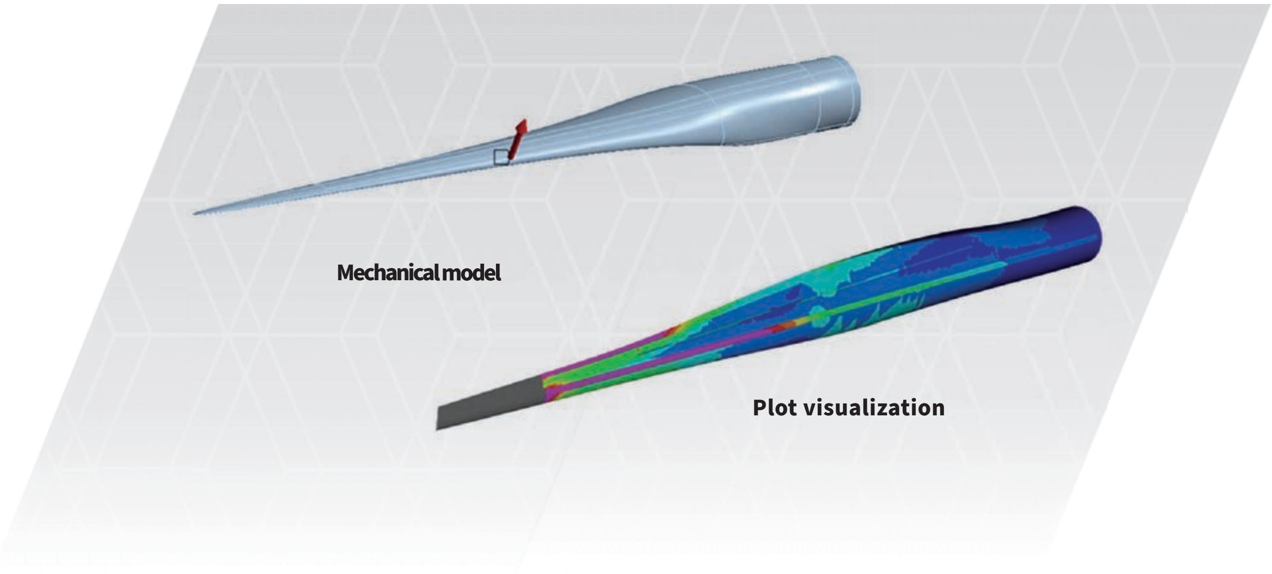
PyAnsys is a suite of open source software that allows you to interact with several Ansys solvers at once within the Python ecosystem, using a programmable interface. This means you can create specialized solutions in Python while integrating Ansys' structural, electromagnetic, and composites simulation solvers (among others), along with other computer-aided engineering (CAE) applications and tools.

The Bewind engineering team uses PyAnsys to create a customized, automated workflow for a more effective and accurate fatigue assessment of their wind turbine blades. This tailored automation saves significantly on time and costs by using the predictive accuracy of Ansys solvers to validate that the blades are both structurally sound and operationally efficient.

TAKE AN AERODYNAMIC LOAD OFF WITH PYANSYS

High-pressure aerodynamic loads are applied to wind turbines to drive rotor blades with enough force to generate mechanical power and, ultimately, electricity. Naturally, this causes vibration during operation. This vibration, coupled with the number of applied loads or stresses, can lead to a host of problems, including crack propagation, delamination, and fatigue phenomena.

Typically, rotor blades are designed with this potential fatigue in mind, yet fatigue loads remain one of the largest contributors to blade damage. Delamination of the composite layers of the blades is the most common cause of total blade failure.



When materials experience fatigue, small cracks initiate and grow under the influence of cyclic loading. Even though the applied load is less than the tensile strength of the material, the repeated, cyclic nature of the loading can cause fracture and failure. Fatigue loads come in two categories: constant and variable amplitude. In real-world scenarios, variable amplitude cyclic loading is more common.

However, because the load's amplitude fluctuates over time, more computer processing is required to simulate damage to the material. The load-time history of parameters such as force, torque, stress, and strain can be used to calculate fatigue cycles. Additional methods can be used to summarize irregular and extended load histories, like the rainflow-counting algorithm. This method is often used to analyze and calculate loading cycles for various amplitudes and then extract a damage parameter using Miner's Rule, one of the most widely used cumulative damage models for fatigue failures. Miner's Rule assumes that the damage done by each stress cycle at any given stress level is equal, which means that the first stress cycle is as damaging as the last.

As you may expect, the computational effort of such algorithms for an entire rotor blade is time-consuming and expensive.

The engineers at Bewind reduce the computational workload by using PyAnsys with open-source, just-in-time (JIT) compilers and parallelization frameworks. These hardware solutions combine with Ansys simulations and other models within Python to accelerate and automate the calculations. The combination of Python with a high-level programming language and JIT compilers generates fast machine code. This enables Bewind engineers to develop state-of-the-art workflows in less time to get the maximum performance out of their workstations. Nowadays, they can analyze all composite layers of an entire rotor blade based on stress time histories in only two days on an HP Z4 computer with 12 threads. Previous workflows took about a week and could only take the top and bottom strain of an element into account.

Additionally, with PyAnsys you can manipulate your simulations to explore even more possibilities in shorter amounts of time. For example, Bewind engineers implemented the invariant-sector method, which accelerates fatigue assessment even more for composite materials. This innovative approach, which relies on a precise selection of areas — or sectors — to evaluate, provides the same level of accuracy as other methods with significantly less computational effort. Fortunately, composite materials show favorable fatigue behavior, so even a slight reduction in exposure to fatigue stress can increase fatigue life. This extended life cycle reduces costs by leveling energy costs over a longer span of time. However, these savings are insubstantial compared to the long-term cost efficiency of ensuring blade reliability by improving fatigue assessment.

EASY AS PYANSYS

By integrating Ansys products into a Python environment, Bewind engineers can customize their fatigue assessment workflow to include standard post-processing tasks and fatigue post-processing of composites. This tailored workflow is achieved by combining packages within the PyAnsys family, Python scripting capabilities, and the customization functionalities of Ansys Composite PrepPost (ACP). This well-crafted approach not only accelerates Bewind's assessment time, but also expands modeling possibilities.

PyAnsys currently includes packages that interface with Ansys Mechanical APDL (MAPDL), a multiphysics simulation and equation solver; Ansys Data Processing Framework (Ansys DPF), a scalable data processing system; and the Ansys Electronics Desktop (AEDT), which includes multiple electronics simulation products to create designs, virtually verify design performance, and implement these designs in large-scale system-level simulations.

PyAnsys offers two options for post-processing within Ansys DPF: PyDPF-Post, which plots data in Ansys binary result files, and PyDPF-Core, which allows you to manipulate your result files and create new data. Further, PyDPF-Core streamlines and simplifies scripting by letting

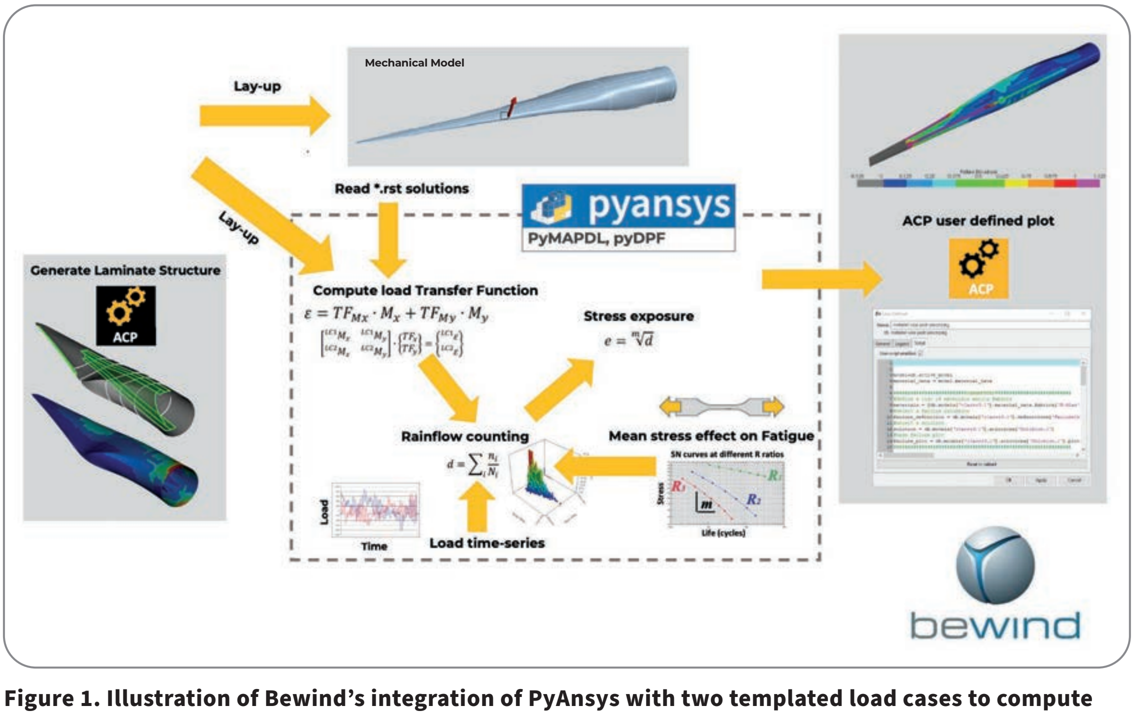


Figure 1. Illustration of Bewind's integration of PyAnsys with two templated load cases to compute load transfer functions at varying stress levels at each composite layer using the binary result files of a rotor blade model.

> ***“As engineers work to make these improvements, they must also ensure the safety of components like the improved blades by verifying their structural integrity and fatigue resistance.”***

you chain, or consolidate, operators and functionalities for a more seamless processing flow. With PyMAPDL, you can create geometry and mesh, plus set up your model. Through PyAEDT, you can simplify scripting for AEDT, which aids in electronics and electromagnetic simulation processing.

In addition to customization, the Bewind team performs advanced analyses for laminated composites, including a successful high-cycle stress-life assessment, which is a post-processing evaluation currently unavailable in standard post-processing software.

As a bonus, Bewind engineers can now access and use Ansys software easily through a standalone native CPython framework outside of Ansys Workbench. The team can select Ansys software from their in-house ecosystem as a regular Python package and combine it with thousands of other available open-source packages within Python. Adding another layer of ease, PyAnsys speaks the language of today’s software developers and users. CPython is the most widely used implementation of the Python language written in Python and C, which blends Python’s object-oriented nature and ease of use with C’s procedural programming style.

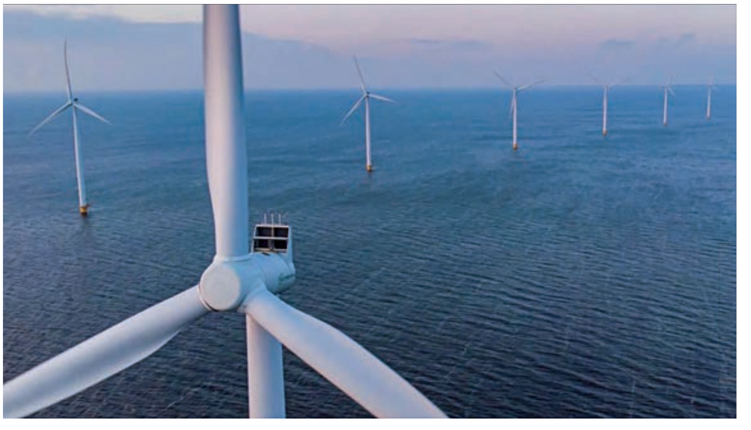
HOW BEWIND SAVES TIME

From multiple Ansys tools at their fingertips to manageable interfaces just clicks away in Python, PyAnsys enables the Bewind engineering team to develop sharper, more compact workflows with less programming effort. Unnecessary read-and-write routines are also minimized, including entering input and output files onto hard disks.

As illustrated in the workflow diagram in Figure 1, Bewind is able to integrate PyAnsys with two templated load cases to compute load transfer functions for stress levels at each composite layer using the result files of rotor blade model simulations.

There are two types of loads to consider when assessing wind turbine fatigue: aerodynamic loads such as shear, drag, and lift in the flapwise, or thrust direction; and inertial loads like gravity or blade dynamics in the lead-lag, or dragging direction. Generally, the flapwise direction is where most loading occurs. Still, both types of loads are leading causes of fatigue because they occur in cyclical, orthogonal bending directions, which create perpendicular intersects that add stress to the blades. In addition, both types of loads have large amplitude

"Our customers take great benefit from this innovative approach, because we can now analyze an entire rotor blade in less than 100 seconds and therefore are much faster to deliver converged blade designs."



and mean value variations, which means there is more load fluctuation to cause fatigue.

As shown in Figure 1, the load transfer function is applied to the load time-series to evaluate the stress time-series in the model. And at Bewind, every load time-series evaluation used in any model is the careful result of hundreds of multibody simulations of the entire wind turbine. Next, each fatigue scenario is identified through a rainflow-counting algorithm accounted for during the design assessment. Following this step, the team uses S-N curves — or stress-life curves — at varying applied load amplitude levels and mean load levels to demonstrate the mean stress effects in a Goodman diagram model. This helps to illustrate the evaluated composite materials and enables the team to determine the amount of damage in each fatigue scenario.

As a final step, the engineers can engage ACP to examine and visualize the overall damage in the laminate via user-defined plots.

LIGHTEN YOUR LOAD, HAVE A SLICE OF PYANSYS

Together with customization, one of the top benefits of working in a Python environment is the access to robust, plentiful Python libraries. You also have access to PyAnsys-specific subroutines supported by efficient and powerful data analytics algorithms to assist with complex studies.

Like Bewind, engineering teams worldwide can use PyAnsys to customize their workflows, accelerate calculations, automate tasks and design processes, power applications, fuel innovation, and more.

With an inventive Python community at hand, an easy-to-use programming language that encourages reuse, and Ansys' leading simulation solutions, PyAnsys delivers nearly endless creative and computational possibilities. To learn more, visit the PyAnsys page on GitHub here: github.com/pyansys.

Improving Valve Performance Under Pressure

By **Dave Converse**, Division Engineering Manager, and **Bipin Kashid**, Senior Simulation Engineer, Parker Hannifin Hydraulic Valve Division, Elyria, Ohio

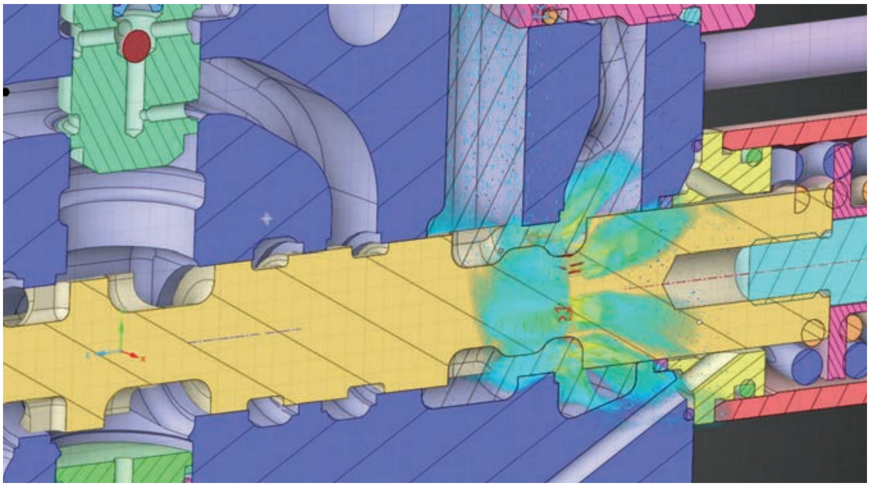
It is often said that a hydraulic valve is the "heart" of the hydraulic system. If there is instability in the system, the issue is rarely the pump, the cylinder, or the hoses. The valve is at the center, so designing an exceptionally performing valve is critical. At Parker Hannifin's Hydraulic Valve Division, simulation-driven design is integral to the product development process, and simulation tools from Ansys are used as one of the key elements to building better valves.

"Parker's evolving use of simulation tools has increased its ability to solve ever more complex problems, and this has led Parker to new customers and opportunities."

The mechanical failures that contributed to the massive Deepwater Horizon oil spill in 2010 focused the world's attention on component reliability, including valves intended to prevent such disasters. Clearly, reliability improvements were needed. While the choke-and-kill systems and blowout preventers used on the rig were based on tried-and-true hydraulic technologies, they simply were not good enough.

For engineering organizations around the globe, the failure of these systems raised questions that needed answers. How can valves intended for use in extreme, dynamic environments be designed to operate more reliably? How can an organization validate that new designs will actually deliver the increased reliability required without many years of prototyping and testing? Behind these questions lie serious design and quality assurance challenges, but equally important are the practical challenges that go hand in hand. How can we reduce the amount of time it takes to build and test prototypes of any new design? How can we minimize the number of design/refinement cycles a more reliable valve will require? And how can we ensure that newer, safer valve designs are affordable in a competitive marketplace?

To assist in addressing some of these questions, Parker engineers rely on pre-validating design concepts through the use of simulation tools across multiphysics domain. Ansys is included as a partner for providing those tools.



Hydraulic valve flow force optimization study

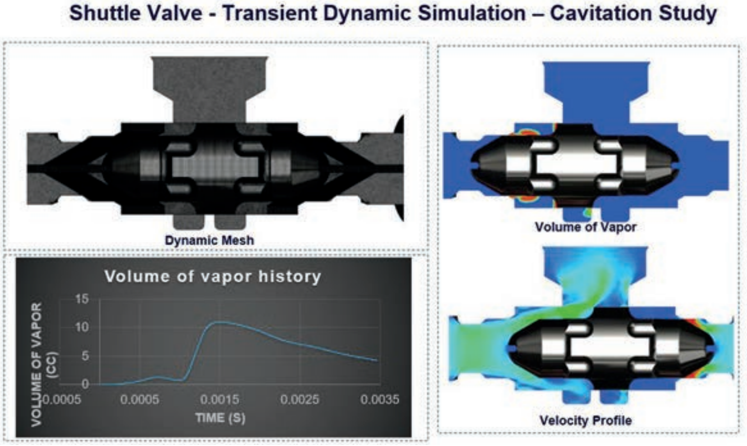
A PLATFORM PRIMED FOR PERFORMANCE

While Elyria, Ohio, is far from any offshore oil rig, it is where Parker works its magic. Here, engineers develop and modify all types of valve designs for customers in industrial markets (including undersea oil and gas operations) and mobile markets — such as companies building heavy equipment for construction, agriculture, forestry, and other industries.

While the requirements of companies seeking solutions for problems that may occur a mile beneath the sea might seem very different from those of companies seeking solutions for a vehicle lifting heavy loads in a logging or mining operation, the underlying valve technologies informing these solutions are not always that different. Parker has a number of hydraulic valve platforms that can be customized and modified to meet a wide range of customer requirements. Simulation plays a decisive role in determining how best to meet those requirements.

In some cases, a clean sheet design approach is utilized. In others, computer-aided design (CAD) models of Parker's stock valves are imported into mechanical simulation tools, such as Ansys Mechanical, where the team can model optimal ways to incorporate specific features or functionality that a customer might require. Modified designs can then be stress- and performance-tested via mechanical simulation to analyze the physical aspects of the valve and its parts, or by using computational fluid dynamics (CFD) simulation tools, such as Ansys Fluent, which analyzes the fluid performance of the designs to determine how each design will operate under different conditions. Based on the results of each analysis, engineers can make refinements to a design in order to optimize valve performance or reliability, and arrive at an ideal combination of materials to balance strength and cost, and more.

Particularly advantageous to Parker is the fact that engineers can design and refine a valve to meet a customer's needs without having to take time to build and test prototypes of each design iteration. Parker's simulation engineers have used simulation tools in conjunction with validation testing for years, thereby building confidence in "first time right" accuracy of the data presented by the simulation tools. Since the old-school process of "design-prototype-test-repeat" could take four to six months to complete — and might have to be completed three, four, or more times — the "design-simulate-refine-repeat-prototype" model of development yields an optimized design in far less time (and often at a far lower cost). Not only will a customer get a prototype faster, but that prototype is far better designed and engineered because simulation has enabled multiple "design-test-refine" iterations before the prototype is manufactured. Only when engineers have a design candidate that optimally delivers the features and functionality that a customer has specified do they construct an actual prototype of the valve.



Subsea valve cavitation study

BETTER VALVES LEAD TO BETTER BUSINESS RELATIONSHIPS

Parker's evolving use of simulation tools has increased its ability to solve ever more complex problems, and this has led Parker to new customers and opportunities. By using simulation to study cavitation erosion in a high-pressure undersea setting, for example, Parker's engineers developed a solution that would dramatically reduce the instabilities inherent in commonly used sub-sea systems caused by cavitation erosion — thereby overcoming a problem that had plagued oil and gas companies for years. New doors opened as a result.

Similarly, by using simulation to study flow forces on a valve used in construction equipment lifting heavy loads, Parker's engineers developed simulation-driven optimization techniques to eliminate valve instability induced by flow forces, which translated to a much smoother, higher-performing vehicle that enhanced an operator's productivity and safety. By gaining greater insight into the effect of flow forces on different components of the valve and running simulations to understand what effects a change to that part's geometry might have, Parker is able to build valves that have proven to be far more stable, reliable, and efficient than its competitors.

And simulation is spreading within Parker. While a specialist team of simulation engineers runs more complex simulations on powerful workstations or a high-performance computing (HPC) system in the cloud, other engineers within Parker are using various simulation tools for simpler problems. A hidden benefit of using many of the same tools across the corporation is the ability to share knowledge, making it easier to share best practices and methods.

In the end, this democratization of simulation delivers benefits that accrue to everyone involved. Engineers at Parker's Hydraulic Valve Division can develop better products faster and at a lower cost, and Parker's customers gain well-engineered hydraulic valves and systems that are tuned to their specifications and perform as expected.

ADDITIONAL RESOURCES:

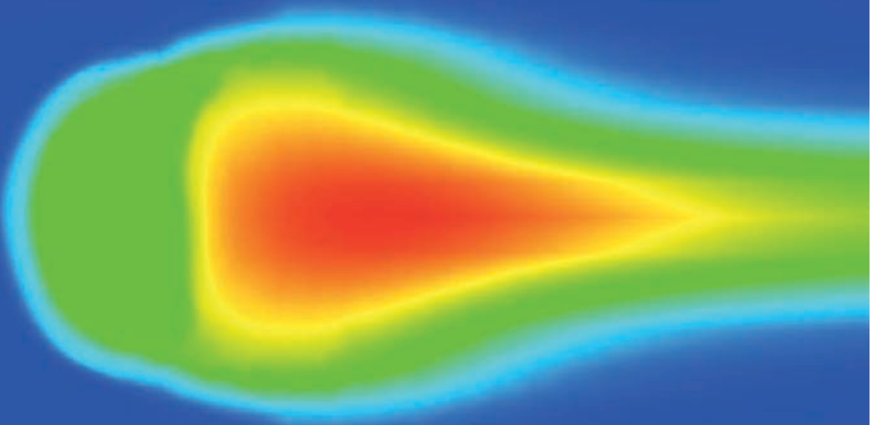
ansys.com/products/structures/ansys-mechanical

ansys.com/products/3d-design/ansys-discovery

ansys.com/products/fluids/ansys-fluent

NanoSUN Delivers Hydrogen Fuel Safely at Scale

By **Jennifer Procario**, Staff Writer, Ansys Advantage



As concerns rise over climate change, carbon dioxide (CO_2) emissions, and other environmental pollutants, industries throughout the world are turning to hydrogen for a solution. A cleaner and safer alternative to fossil fuels, hydrogen fuel offers near-zero CO_2 emissions when produced using renewable energy sources such as the electrolysis of water to release hydrogen or solar-based processing. While its production may vary, the applicability and benefit of "green" hydrogen remains the same across industries and private sectors — from the aerospace and automotive industries, to construction companies, and beyond.

With such value to the environment and use in various fields, it is critical that clean hydrogen fuel is easily accessible and properly stored, distributed, and dispensed. At least, that's the top priority at NanoSUN, an award-winning engineering company based in the United Kingdom dedicated to the development, manufacturing, and commercialization of green hydrogen refueling solutions.

As a member of the Ansys Startup Program, NanoSUN is applying Ansys Fluent computational fluid dynamics (CFD) simulation software to make this happen more safely, efficiently, and at scale.

“I think I’d quite like to have a look at more detail, using Ansys Fluent to model the flow within the system, instead of external to the system if there’s a release. It would be very interesting to model flow into the cylinders and through orifices, such as how it reacts in plug flow.”

GETTING AHEAD WITH ANSYS

Determined to bridge the gap between hydrogen suppliers and fuel-cell users, industry experts from the industrial gas and hydrogen fuel-cell sectors came together in 2017 to establish NanoSUN. The startup’s driving mission is to develop high-pressure, mobile, and cost-effective hydrogen storage, distribution, and dispensing solutions to the transport and mobility industries.

Still, hydrogen storage and its delivery present key challenges involving weight, volume, efficiency, safety, and cost. Further, government agencies and regulatory associations administer safety standards regarding the handling of hydrogen due to its high flammability coupled with its lower ignition energy. Essentially, it can ignite faster than gasoline or natural gas.

As a new company, resources at NanoSUN were limited. To meet challenges, adhere to safety regulations, and thrive in the industry, the team needed access to sophisticated engineering simulation software and tools. NanoSUN learned about the Ansys Startup Program through Ansys’ U.K. Channel Partner, Wilde Analysis Ltd, and became a member in early 2021. The program provides early-stage startups with affordable access to Ansys’ extensive portfolio of simulation solutions and support to help advance their businesses.

“It’s very important for us to have access to Ansys simulation software,” says Bethany Ladd, a graduate project engineer at NanoSUN. “It gives us the ability to create models, which gives us a better understanding of what we can expect from our systems, and it enables us to ensure that we’re creating a safe environment for the customer. Being a small company, until the past year and a half we haven’t had the funding for a proper commercial license, so the ability to jump on the Ansys Startup Program specifically gave us access where previously we wouldn’t have been able to afford these tools.”

Safety is key. With cylinders that hold around 419 kilograms of hydrogen at a rated pressure of 425 bar, fire is an active threat if appropriate ventilation is not in place or properly functioning.

This is where Ladd and her team apply Ansys Fluent the most, tracking and monitoring potential leaks within the refueling station’s venting system to prevent fire. Routine system checks are especially important at NanoSUN, where the refueling process engages multiple cylinders at once.

Most refueling companies use decant technology, which involves high pressure flowing to low pressure, from one cylinder to another, stopping when both are balanced. This state is referred to as equalization. However, this technique only partially fills the receiving cylinder.

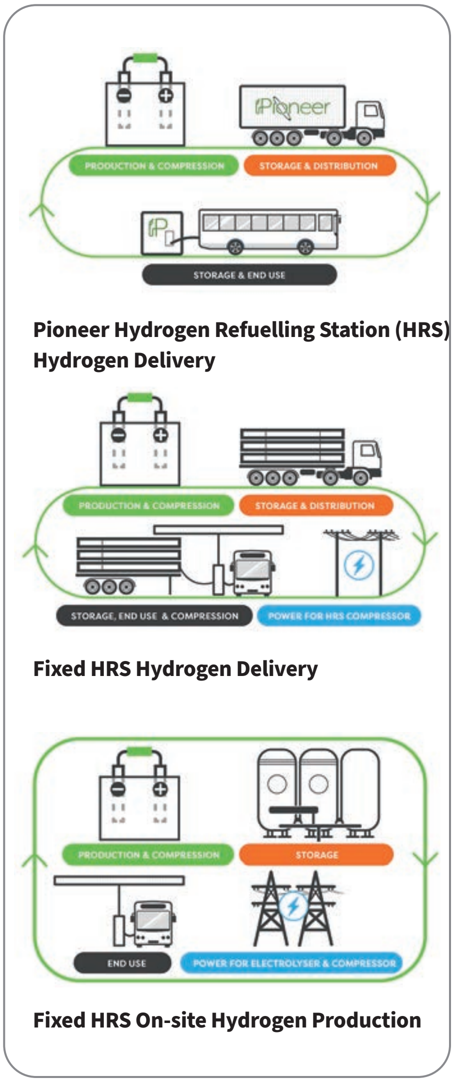


Figure 1: This diagram illustrates the advantages of NanoSUN’s mobile Pioneer Hydrogen Refuelling Station (HRS) solutions (top) in comparison to fixed HRS solutions (center and bottom).

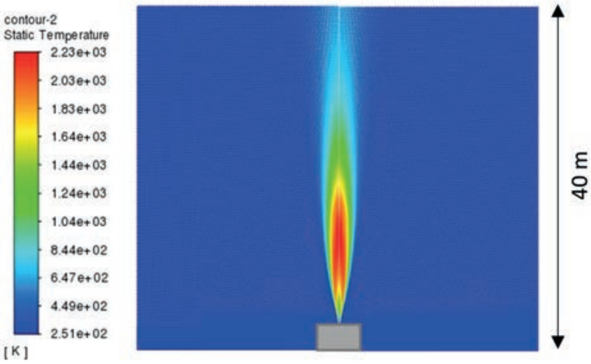


Figure 2: Temperature profile illustrates the thermal release of nine cylinders.

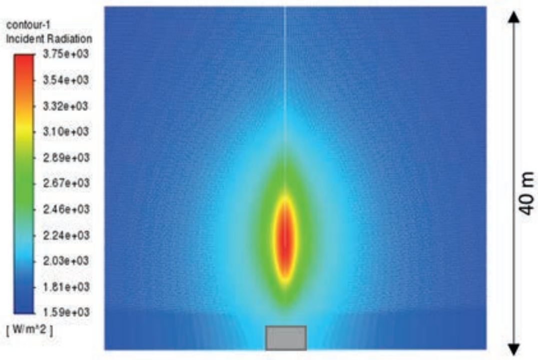


Figure 3: Radiation profile illustrates the thermal release of nine cylinders.

Alternatively, NanoSUN employs its unique Pioneer Cascade Technology for refueling, which consists of an automated system of individual decants. This sophisticated system uses multiple refueling cylinders, which completely fill the receiving cylinder. Both the decant method and the cascade technology use the pressure gradient between a higher-pressure source and a lower-pressure target tank to transport hydrogen fuel without an external energy source or compressor. However, NanoSUN's cascade method enables rapid, low-cost deployment of hydrogen to make it more accessible and scalable.

NanoSUN's signature Pioneer Hydrogen Refuelling Station (HRS) is a fully mobile, automated refueling solution that delivers hydrogen fuel directly to the point of use, facilitating more efficient and timely distribution (see Figure 1). With NanoSUN's cascading system built in the form factor of a standard 20-foot shipping container, the Pioneer HRS is a one-stop shop to meet a client's hydrogen refueling needs. The only thing needed to operate on site is a grounding connection and 230 V power supply. Also, each unit is equipped with a battery backup.

Its innovative and convenient transport makes it available and suitable for a wide range of applications, including hydrogen buses, vans, and trucks; material handling; and construction equipment. It can also be used as a backup solution to traditional hydrogen fixed stations, for fleet demos, or as a network extender.

For vehicle fleets particularly, NanoSUN's Pioneer HRS offers significant time and cost savings by eliminating the need to power additional compressors or electrolyzers at other stops.

STAYING AHEAD: SAFETY FIRST WITH ANSYS FLUENT

Joining NanoSUN just six months ago, Ladd has already made an impact thanks to her experience using Ansys' CFD software through the Ansys Academic Program as a student at the University of Cambridge in England. In this program, Ansys provides universities with discounted software while offering students free resources for self-learning, a combination that gives aspiring engineers an edge after graduation.

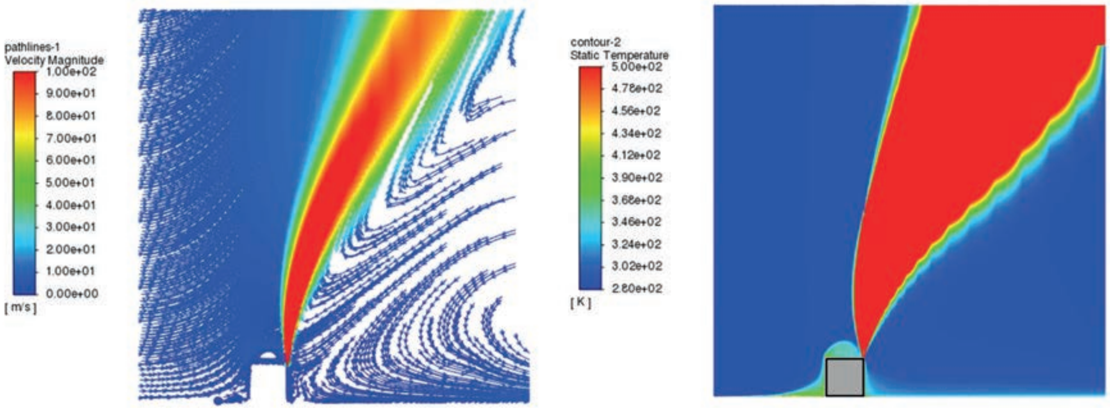


Figure 4: Velocity colored streamlines and temperature contours for a 2D planar thermal pressure release device (TPRD) model with 5 m/s wind coming from the left

"When I started at NanoSUN, I wasn't aware that they were intending to use CFD simulation. So, it was really interesting to be able to bring in what I had done at university through the Ansys Academic Program," says Ladd.

The bulk of Ladd's role is to examine the engineering and safety calculations for NanoSUN's refuelers, using Fluent to model scenarios such as hydrogen releases, or leaks, and potential combustion points throughout all stages of development. Since hydrogen is the lightest element and disperses at such a rapid pace, Ladd and her team need the right tools to thoroughly observe its behavior.

"We've been using Ansys Fluent for complex releases," Ladd says. "With hydrogen being quite a buoyant gas, we use Ansys Fluent to look at things like if we were to have a leak into our container, how would we expect the flow to move around the other cylinders? I don't think you could model these things without CFD."

Additionally, Ladd analyzes potential combustion points to understand and gauge the direction and magnitude of potential releases or flames to calculate proper hazard distances, adhering to related hydrogen safety standards.

Some safe distances for hydrogen, as outlined by the British Compressed Gas Association, include 5 meters from flammable gas storage, sources of ignition, and fuel gas vent pipes; and 8 meters from employee offices or congregations of people, flammable liquids, and ventilator air intakes.

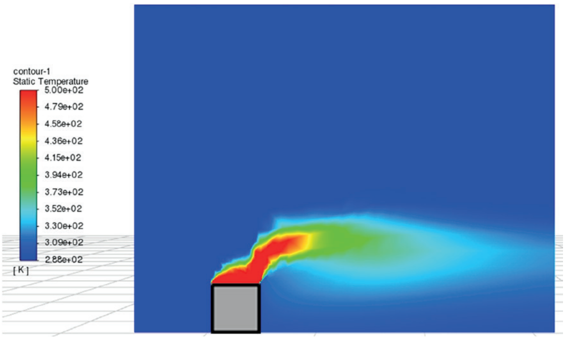


Figure 5: Temperature contours across the central plane of the model (through the hydrogen release point) for a 3D model with incident wind speed of 20 m/s from the left

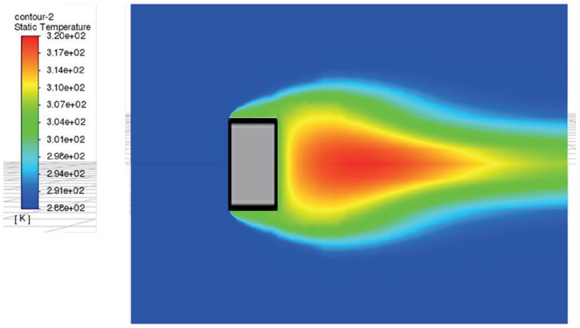


Figure 6: Temperature contours across a horizontal plane of constant height 1.8 meters. In this model, the wind at 20 meters per second is coming from the left side of the graphic.

Such precautions are critical to Ladd's work and consider variables such as temperature and radiation.

An example of this is when Ladd and her team use Fluent to model thermal pressure relief devices (TPRDs) to anticipate the temperature at which a fire may erupt due to a gas release.

Using heat radiation guidelines given in thermal dosage from the European Industrial Gases Association (EIGA), Ladd applies Fluent to calculate horizontal distances between the point of hydrogen release and people working nearby. These thermal measurements incorporate dosage for various probabilities of injury, exposure durations, and thermal radiation intensities. With the radiation models built into Fluent, Ladd and her team compute the radiation incident at different distances from the release using the EIGA data and apply a safety factor of 50% to determine the minimum safe distance from the release.

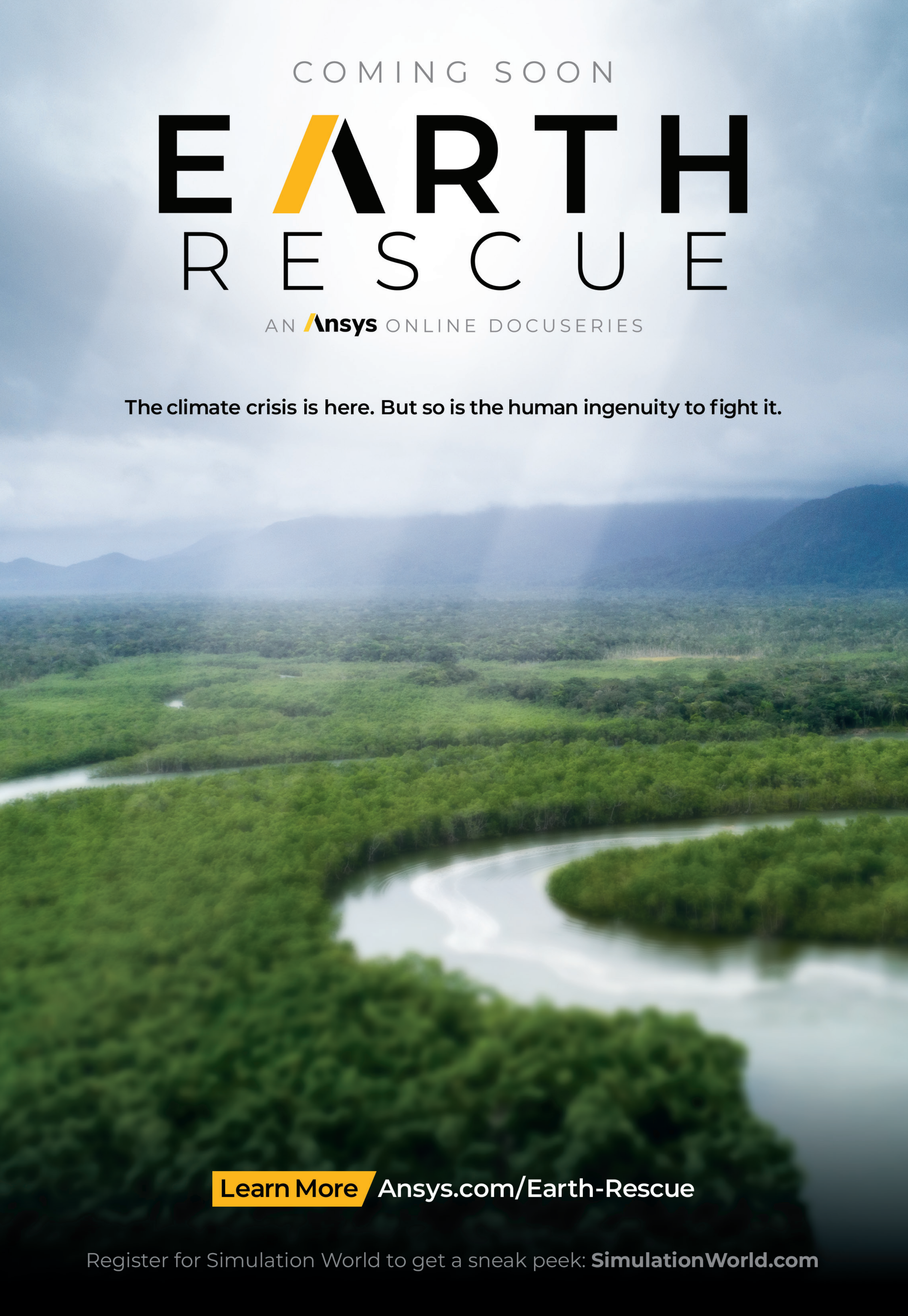
Through these analyses, the engineers are able to predict, monitor, and prevent catastrophe.

For example, the team learned that if nine cylinders are released at once and the release is ignited, the radiative hazard distance is 19 meters. Alternatively, if only one cylinder is released through a TPRD, the distance would only be 1.5 meters. Figures 2 and 3 show the outputs for a model of nine cylinders being released through the TPRD system assuming ignition.

NanoSUN engineers also model the convective heat of TPRD releases, integrating wind at different speeds (shown in Figure 4) with 2D models and accounting for turbulence around the container with 3D models (shown in Figure 5). Further, the team observes temperature contours across horizontal planes at a constant height of 1.8 meters (shown in Figure 6), as recommended by the EIGA as the point of flammable effects at the average "head height."

Beyond these capabilities, Fluent offers multiphase flow analysis and combustion modeling, providing insight into system performance and flow phenomena. Ladd plans to explore these next, taking full advantage of Ansys software and program offerings.

"I think I'd quite like to have a look at more detail, using Ansys Fluent to model the flow within the system, instead of external to the system if there's a release," says Ladd. "It would be very interesting to model flow into the cylinders and through orifices, such as how it reacts in plug flow."


COMING SOON
EARTH
RESCUE
AN Ansys ONLINE DOCUSERIES
The climate crisis is here. But so is the human ingenuity to fight it.
Learn More Ansys.com/Earth-Rescue
Register for Simulation World to get a sneak peek: SimulationWorld.com